AutoSketch®

autodesk®

Getting Started

9

June 2004

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Contents

Installation

This chapter tells you how to install AutoSketch®on your computer. After you install the software, view the *Readme* (click Read Me on the Help menu). The *Readme* contains important information that was compiled after this guide was printed.

For step-by-step instructions about learning the product, read the entire *Getting Started* guide. You can find a PDF version of this guide in the following location ("C" is the installation drive letter):

C:\Program Files\Autodesk\AutoSketch9

In this chapter

Contents of the AutoSketch Package

The AutoSketch package includes the following:

- AutoSketch CD
- AutoSketch *Getting Started* (this guide)

System Requirements

Before you install AutoSketch, make sure that your computer meets the minimum system requirements.

Hardware and software requirements

Hardware/Software	Requirement	Notes
Operating system	Windows®XP Professional Windows®XP Home Windows®2000	
Web browser	Microsoft®Internet Explorer 6.0 with Service Pack 1 (or later)	
Processor	Pentium III or later 300 Mhz	
RAM	128 MB	
Video	800 x 600 (minimum) 1024 x 768 with 64K colors (recommended)	Requires a Windows-supported display adapter.
Hard disk	180 MB	
Pointing device	Mouse, trackball, or other device	
CD-ROM	Any speed (for installation only)	
Optional hardware	Graphics card 16 MB (minimum) Printer or plotter Modem or access to an Internet connection	

Install AutoSketch

This section explains how to install AutoSketch on a stand-alone computer.

To install AutoSketch

1. Insert the AutoSketch CD into your CD-ROM drive.
2. Follow the installation prompts that are displayed.

If installation did not begin when you inserted the AutoSketch CD into your CD-ROM drive, Autorun may be turned off on your machine.

To install AutoSketch if Autorun is turned off

1. Insert the AutoSketch CD into your CD-ROM drive.
2. On the Start menu (Windows), click Run.
3. In the Run dialog box, enter <***CD drive letter***>:**Setup.exe** and click OK.
4. Follow the installation prompts.

Congratulations! You have successfully installed AutoSketch. You are now ready to register your product and start using the program.

Register AutoSketch

Registering AutoSketch makes you eligible for technical support and for early notification of new product releases.

To register AutoSketch

1. On the Start menu (Windows), click All Programs (or Programs) ➤ Autodesk AutoSketch 9.
2. In the AutoSketch Product Registration wizard, select Register Now, and then click Next.
3. Follow the on-screen instructions.

Add or Remove Features

You can add and remove features in AutoSketch at any time. For example, if you chose a Custom installation when you first installed AutoSketch, you you can add features not originally installed.

To add or remove features

1. In the Control Panel (Windows), start Add or Remove Programs.
2. In the Add or Remove Programs window, click AutoSketch Release 9, and then click Change.
3. In the Setup wizard, Application Maintenance page, select the Modify option, and then click Next.
4. On the Select Features page, select the features you want to add or remove, and then select one of the following options:

   - **Will Be Installed on Local Hard Drive.** Installs a feature or components of a feature on your hard drive.
   - **Entire Feature Will Be Installed on Local Hard Drive.**
   - **Entire Feature Will Be Unavailable.** Removes the feature.

   Note To revert to the AutoSketch features selected in your original installation, click Reset.

5. Click Next.
6. On the Content Unit page, select one of the following, and then click Next.

   - U.S. (Imperial) Content
   - Metric Content

7. On the Ready to Modify the Application page, click Install.
8. In the AutoSketch dialog box, click Finish.
9. If prompted, restart your computer.

Repair AutoSketch

If you accidentally delete or alter files that are required by AutoSketch, it might not perform correctly. You can attempt to repair AutoSketch.

To repair AutoSketch

1. In the Control Panel (Windows), start Add or Remove Programs.
2. In the Add or Remove Programs window, select AutoSketch Release 9, and then click Change.
3. In the Setup wizard, Application Maintenance page, select the Repair option, and then click Next.
4. On the Ready to Repair the Application page, click Finish.
5. If prompted, restart your computer.

Uninstall AutoSketch

When you uninstall AutoSketch, all components are removed from the computer.

Note Uninstalling the application does not automatically delete drawing files you have created. You can delete those files manually.

To uninstall AutoSketch

1. In the Control Panel (Windows), start Add or Remove Programs.
2. In the Add or Remove Programs window, select AutoSketch Release 9, and then click Remove.
3. In the message box that is displayed, click Yes to remove AutoSketch.
4. If prompted, restart your computer.

Make the Transition from Paper to CAD

2

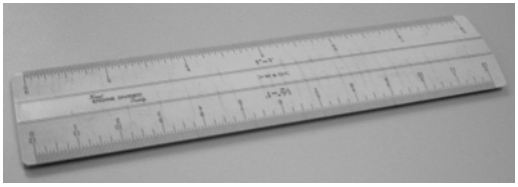
With your decision to use AutoSketch®, you have entered the world of computer-aided design (CAD). AutoSketch makes your drawings more precise and you more productive than you have been using paper as your design format.

This chapter explains how you can take your drafting knowledge and apply it to CAD.

In this chapter

Draw to Scale

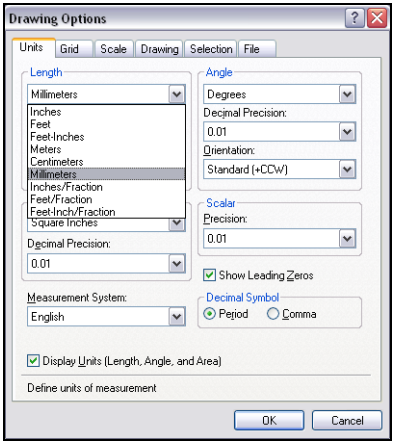
Drawing scale is something you consider when laying out your drawing. You establish scale differently in CAD than you do with manual drafting.



Draw the object at full scale in the units you specify.

With manual drafting, you must determine the scale of a view before you start drawing. This scale compares the size of the actual object to the size of the object drawn on paper.

In AutoSketch, drawings are created using the real-world values you specify. The computer handles scaling the drawing to fit on paper.



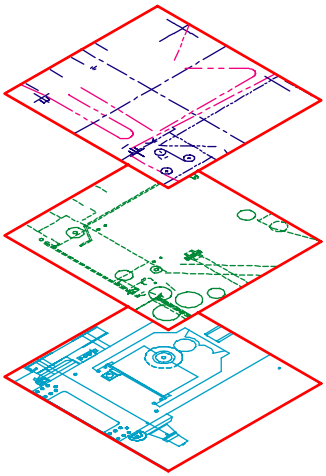
When you lay out and plot your drawing, you can set any scale you like.

For example, you can use feet and inches, or meters and kilometers, and so on. You might draw a motor part with millimeters as the unit of length so that entering **25** means "25 millimeters." When you draw a map, you might select kilometers so that entering **25** means "25 kilometers."

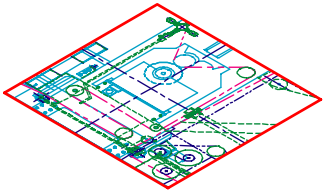
Although you can easily change scaling at any point while drawing, it is useful at the start to select a scale that is appropriate to the drawing you are working on. This allows you to keep your drawing on the "page" that AutoSketch displays onscreen.

Organize Drawing Information

In both manual drafting and CAD, you need a way to organize your drawing content—a method for separating, sorting, and editing specific drawing data.



Turn off layers to hide complex details as you work.



Display layers when you need to see all components.

With manual drafting, you can separate information onto individual transparent overlays. For example, a building plan might contain separate overlays for its structural, electrical, and plumbing components.

In AutoSketch, *layers* are equivalent to transparent overlays. As with overlays, you can display, edit, and print layers separately or in combination.

You can name layers to help track content, and lock layers so they can't be altered. Assigning settings such as color, pen style, or pen width to layers helps you comply with industry standards.

You can also use layers to organize drawing objects (called *entities* in AutoSketch) for printing.

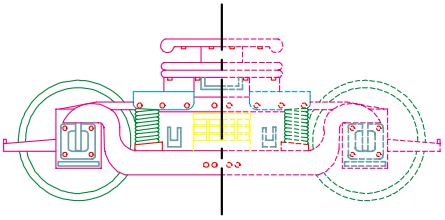
This mechanical drawing of a press uses layers to show different types of information in different styles and colors.

Draw Efficiently

Draw with less effort and revise with more speed: these are the two main reasons you use CAD. AutoSketch has a complete set of drawing and editing tools to help eliminate repetitive, time-consuming drafting tasks.

If you work with paper and a drawing board, your set of drawing tools is likely to include pencils, scales, parallel rules, templates, and erasers. Repetitive drawing and editing tasks must be done manually.

In AutoSketch, you can choose from a variety of drawing tools that create lines, rectangles, circles, curves, and more.



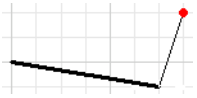
You can save drafting time by drawing one half of an item and then mirroring it to create the other half.

With AutoSketch, you can easily copy, scale, rotate, and mirror entities. You can move or copy entities between open drawings or within the same drawing. Editing is easy with tools such as stretch, align, and offset. To add hatching, simply insert a hatch pattern from the AutoSketch *Content Librarian* into the area to be filled.

In this drawing of a trolley, copying and mirroring were used to create repeated and symmetrical features. Offsetting lines and hatching were also used to draw more efficiently.

Draw Accurately

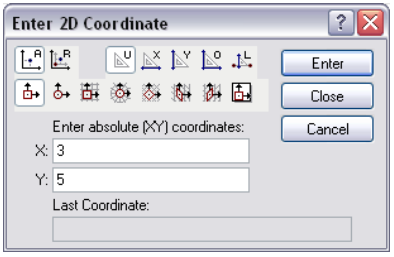
Engineering and architectural drawing require a high degree of accuracy. With CAD, you draft more accurately than with manual methods.



Snaps allows you to draw with precision.

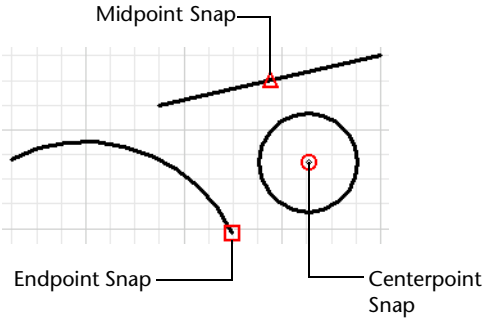
On paper, you must draw objects carefully to ensure correct size and alignment. Objects drawn to scale must be manually verified and dimensioned.

In AutoSketch, you can ensure exact dimensions by using several methods. The simplest method is to locate points by snapping to some interval of a *grid*.



Another method is to specify exact *coordinates*. Coordinates specify a drawing location by indicating a point along an X and Y axis or a distance and angle from another point. You can specify coordinates that are relative to other points or to the drawing's coordinate system.

You can also snap to locations on existing entities, such as an endpoint of an arc, the midpoint of a line, or the centerpoint of a circle.



View Your Drawing

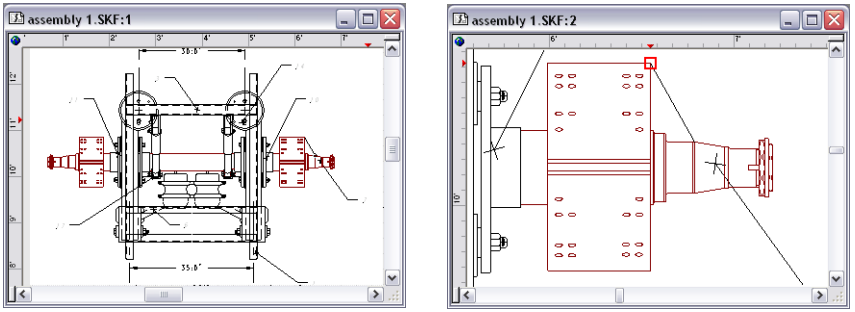
The power of CAD makes it easy for you to quickly view different parts of your design at different magnifications.

When you draft on paper and need to work on another section, you must physically move to that area of your drawing.

In CAD, the size and resolution of your computer monitor limit your viewing area. AutoSketch viewing methods bypass this limitation.

To do detailed work, you can increase display size by *zooming* in. You can zoom out to display more of the drawing. To move to another section of a drawing, you *pan* the drawing without changing magnification.

You can view several areas of your drawing simultaneously by creating additional *windows*. Windows can be arranged automatically or manually. They let you work easily on different parts of your drawing. Changes in one window are reflected in the others.



Windows display different portions of your drawing simultaneously. You can zoom and pan the display in each window independently

With *detail views,* you can arrange additional views at different zoom levels or scales. You can create split windows, and you can pan and zoom in each window to create the best working conditions.

Modify Your Drawing

Revisions are a part of any drawing project. Whether you work on paper or with CAD, you will need to modify your drawing in some way.

If you stretch an entity...

...the hatch adjusts automatically.

On paper, you must manually erase and redraw to make revisions to your drawing.

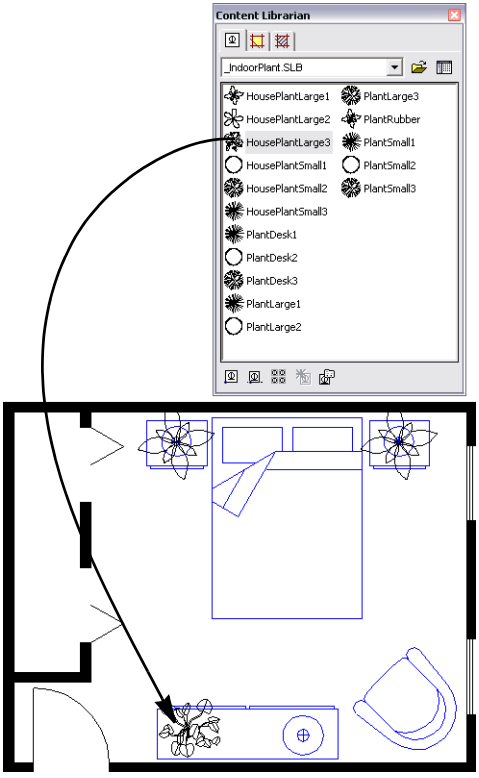
AutoSketch eliminates tedious manual editing by providing a wealth of editing tools. If you need to copy all or part of an entity, you don't have to redraw it. If you need to remove an entity, you can erase it with a few clicks of the mouse. And if you make an error, you can quickly undo your actions.

Once you draw an entity, you never need to redraw it. You can modify existing entities by mirroring, rotating, scaling, stretching, trimming, and more. At any time, you can change entity properties, such as pen style, pen width, color, and layer.

These before-and-after drawings show some typical edits to a house elevation.

Use Standard Symbols

Symbols have long been used in manual drafting as a way to represent real-world objects in a simplified way. The ability to create and reuse standard symbols is one of CAD's greatest strengths.



The Content Librarian lets you locate libraries (collections) of symbols. You choose the symbol you want and drag it into your drawing.

With manual drafting, you might use a symbol template or printed stickers to draw repetitive landscape, architectural, mechanical, or electrical symbols. This method, however, limits the possible variations of a symbol.

In AutoSketch, you can save time by inserting symbols from the Content Librarian anywhere in your drawing, at any rotation or scale.

You can then add a symbol as many times as needed by simply clicking to place the symbol.

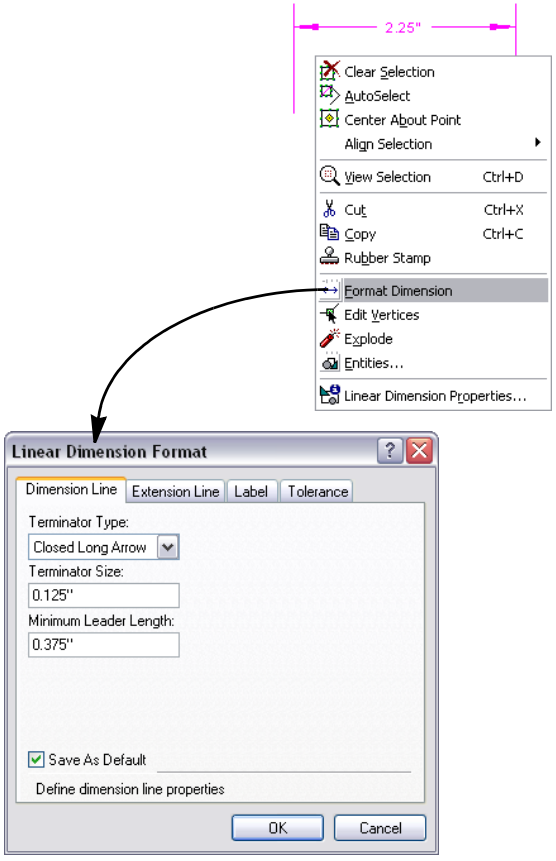
Should a standard symbol change (be *redefined*), all instances of the symbol in your drawing will automatically be replaced.

In AutoSketch, you can also create your own symbols from scratch or modify existing ones.

Standard landscaping symbols are used in this drawing of a residential home floor plan.

Create Dimensions and Text

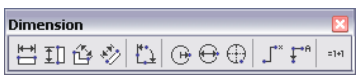
Creating accurate dimensions and consistent, legible text is a time-consuming task for the manual drafter. CAD provides ways to streamline this task.



When you work on paper, you typically draw to scale and then add dimensions and annotations. If you resize any part of the drawing, you must erase and then redraw the dimensions. Changing text can often involve relettering the whole drawing. AutoSketch automates the process of creating and changing dimensions and text.

In AutoSketch, you can customize individual dimensions, and when information changes, you can easily revise text, including its content, font, height, angle, and justification.

Virtually all standard dimensioning types are provided in AutoSketch: linear, radial, ordinate, angular, baseline, and more.



AutoSketch Basics

3

In this chapter, you learn how to start AutoSketch® and use the Startup dialog box to create or open a drawing. You also learn about the user interface and the basic features and functionality of AutoSketch. Once you have learned these AutoSketch "basics," you can do the exercises in this guide's tutorials and learn to use the product.

More information about each of these components and features is available in the Help system.

In this chapter

Introduction

AutoSketch is a precision drawing tool for the Microsoft® Windows® XP and Windows® 2000 operating systems. The emphasis in AutoSketch is on speed, power, and ease of use.

AutoSketch features appear when you need them, and are kept out of the way when you don't. If you're already a Windows 2000 or Windows XP user, you'll find the menu system and much of the user interface familiar. If you're new to Windows, you'll find AutoSketch an easy place to work.

In this chapter, you learn how to start AutoSketch and use the Start Up dialog box to create or open a drawing, and you get familiar with the user interface components.

Start AutoSketch

When you start AutoSketch, you can begin a new drawing, start with a template, or open existing drawings. Simply click a selection and begin.

To start AutoSketch for the first time

- On the Start menu (Windows), click All Programs (or Programs) ➤ Autodesk AutoSketch 9.

The AutoSketch Start Up dialog box is displayed.

Use the Start Up Dialog Box to Create or Open a Drawing

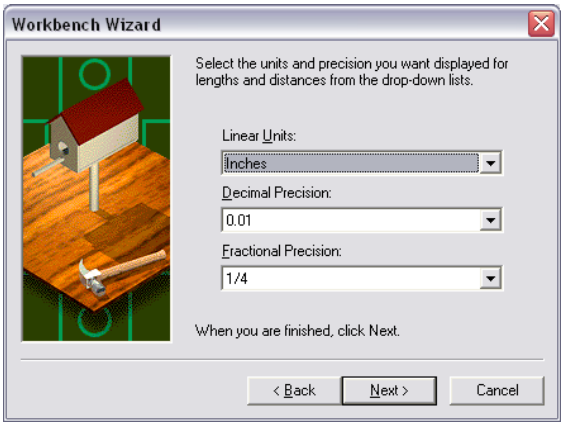
The Start Up dialog box has three tabs with options for starting a drawing:

- **Wizard tab.** Allows you to start a drawing immediately or choose one of the listed wizards.
- **Template tab.** Allows you to base a drawing on a template, and to preview and organize the templates.
- **Open tab.** Allows you to open a recently used file, browse for a file, and preview a selected file.

Start a Drawing or Choose a Wizard (Wizard Tab)

In the Start Up dialog box, Wizard tab, you can start a drawing immediately or choose a wizard to start a drawing.

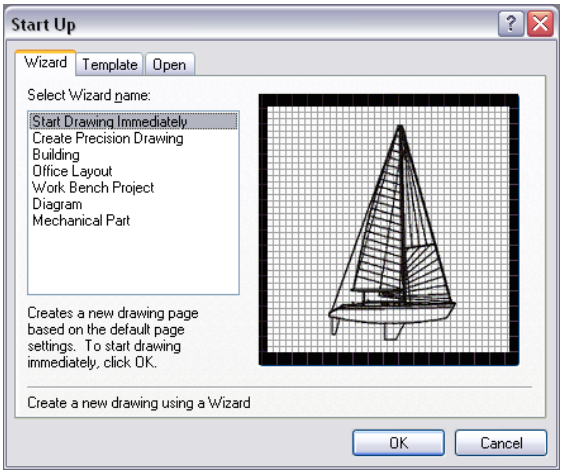
A wizard contains instructions to guide you through the steps to accomplish a task. The AutoSketch Start Up wizards help you make drawing decisions to set up a drawing. If you are drawing a workbench project, for example, the wizard steps you through logical workbench decisions for that drawing.

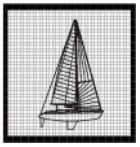


Example of a page in the Workbench wizard

Tip During an AutoSketch work session, you can access wizards by clicking File ➤ New.

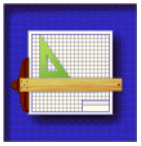
The illustration shows the Wizard tab, and is followed by an explanation of each of its choices.



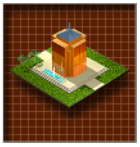


Start a Drawing Immediately. Creates a new drawing based on preset settings such as page size and scale. You can always change these settings later. Select this option, and then click OK. You are ready to begin drawing.

The following choices on the tab are wizards:



Create Precision Drawing. Creates a new drawing based on settings that you specify. You enter summary information (including drawing title, project name, and so on), drawing size and scale, units of measurement, and grid spacing



Building. Sets up a drawing of a commercial building, home, or exterior site layout. You choose the building shell, dimensions, wall thickness, roof generation, database report types and fields, layers, page orientation, and useful toolbars. You can add symbols such as telephone poles, trees, and hydrants.

Office Layout. Sets up a drawing of a single office or an entire floor of offices. You choose the office shell, dimensions, wall thickness, database report types and fields, layers, page orientation, grid settings, and useful toolbars. You can add symbols such as telephones and computer equipment.

Work Bench Project. Sets up a drawing of a small home, or a mechanical or woodworking project. You specify page orientation, units and precision, scale, grid options, database report types and fields, layers, and useful toolbars.

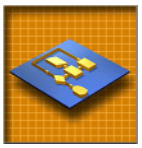


Diagram. Sets up a diagram such as organizational tree, Web site map, flow chart/schedule, piping, electronic schematic, logic diagram, networking, or PC board layout. You choose the type of diagram, page orientation, and useful toolbars.

You can add symbols such as flowcharts, schedules, piping, switches, capacitors, lamps, switchboxes, PCs, printers, mainframes, modems, circuit chips, soldering points, and so on.

Mechanical Part. Sets up a drawing of a small machine or machine component. You specify page orientation, units, precision, scale, annotation options, Edit command settings, grid options, page division (for different views of a part), database reports and fields, layers, and useful toolbars. You can add symbols such as nuts, bolts, screws, brackets, washers, and so on.

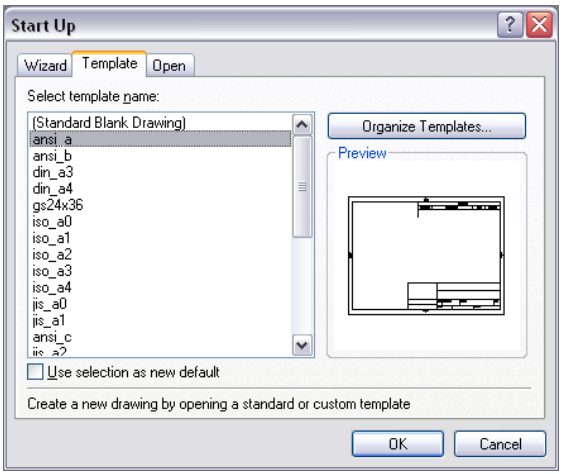
Choose a Template (Template Tab)

In the Start Up dialog box, Template tab, you can choose a template file to start a drawing.

AutoSketch includes dozens of drawing template files. A template is a drawing file that has settings such as borders, title blocks, grid spacing, drawing scale, and page size already selected for you. When you select one of the templates in the list, you can preview it in the Preview area. Then, you simply choose the template that is right for your project.

You can also create your own template from an existing drawing. If you create the same type of drawing each time you work with AutoSketch, you may want to redefine the default template by saving an existing drawing as a template, and then selecting that template as the new default. Then, you can use the template to create new drawings of the same type.

The following illustration shows the Template tab.

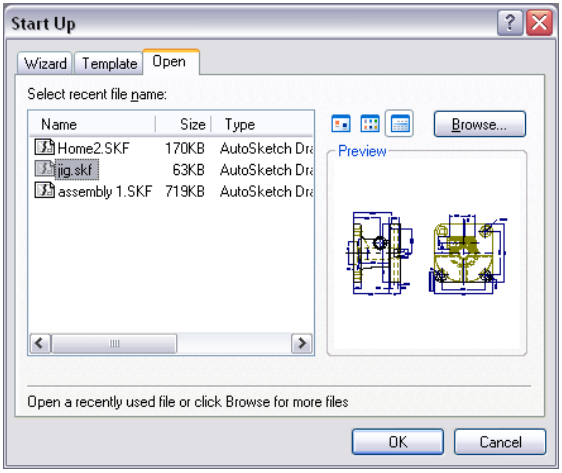


Tip During an AutoSketch work session, you can access templates by clicking File ➤ New.

Open an Existing Drawing (Open Tab)

In the Start Up dialog box, Open tab, you can open an existing drawing file. You can adjust how files are displayed in the list, browse for more files, and preview a selected file.

The following illustration shows the Open tab.



Tip During an AutoSketch work session, you can access existing drawings by clicking File ➤ Open.

Understand the User Interface

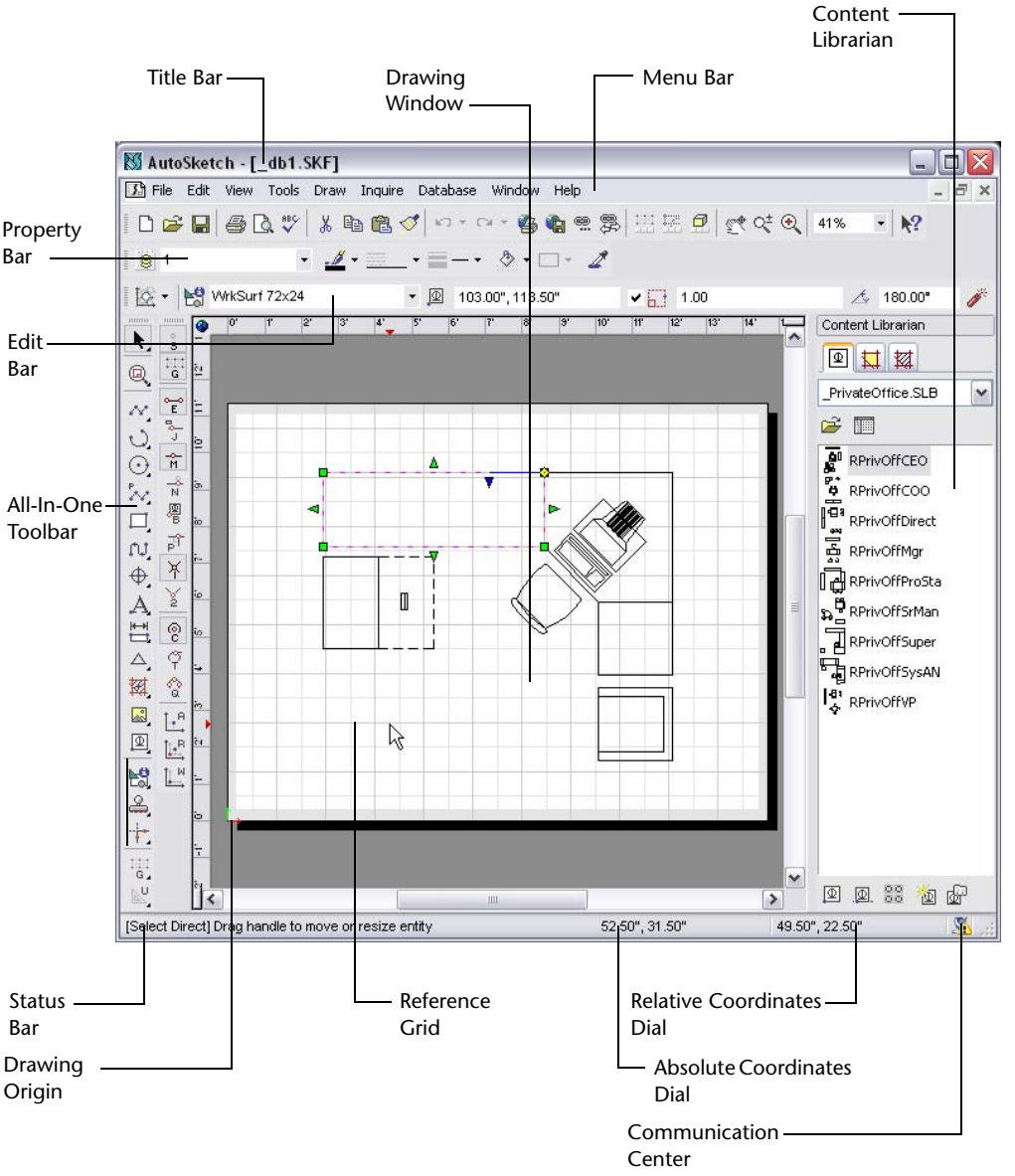
The first step in learning how to use AutoSketch is to become familiar with its user interface.

Drawing Window

Once you choose the type of drawing you want to create, AutoSketch opens a drawing window. The drawing window in AutoSketch is the space where you work.

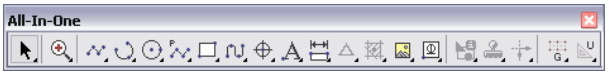
Many drawing windows can be open at one time. Clicking a drawing window makes it active so that you can work in it. You can make changes in the active window only. You can resize, minimize, maximize, and close each drawing window independently.

The following illustration shows the AutoSketch user interface elements in a drawing window. These elements are listed alphabetically and described after the illustration.


Content Librarian
Title Bar
Drawing Window
Menu Bar
Property Bar
Edit Bar
All-In-One Toolbar
Status Bar
Reference Grid
Relative Coordinates Dial
Drawing Origin
Absolute Coordinates Dial
Communication Center
AutoSketch - [_db1.SKF]
File Edit View Tools Draw Inquire Database Window Help
WrkSurf 72x24
103.00", 113.50"
1.00
180.00"
41%
Content Librarian
_PrivateOffice.SLB
RPrivOffCEO
RPrivOffCOO
RPrivOffDirect
RPrivOffMgr
RPrivOffProSta
RPrivOffSrMan
RPrivOffSuper
RPrivOffSysAN
RPrivOffVP
[Select Direct] Drag handle to move or resize entity
52.50", 31.50"
49.50", 22.50"

All-In-One Toolbar

As its name suggests, the All-In-One toolbar contains buttons that help you perform most of the tasks that you need to do to create a drawing. Take a few moments to understand how this toolbar works before doing the exercises in this guide.



Most of the buttons on the AutoSketch specialized toolbars can be found on the All-In-One Toolbar. For example, the All-In-One toolbar contains all of the 15 snap tools on the Snap toolbar. Simply click and hold the Snap to Grid button on the All-In-One toolbar, and the other Snap tools are displayed on the toolbar that appears, called the *toolset*. To display a tooltip for any toolbar button on a toolset, keep the mouse button depressed and place your pointer over the button. To select a button, release the mouse button.

In the exercises in this guide, you are instructed to use the Draw menu, the All-In-One toolbar, and other toolbars to complete the tasks. You may find that many of the same tasks can be performed by using different toolbars included in AutoSketch. You can learn more about those toolbars in the Help system.

Note The following table shows the buttons that are displayed on the All-In-One toolbar when you first use AutoSketch. When you click a button on a toolset, that button "sticks," or remains the active button, until you click a different button on the same toolset.

All-In-One Toolbar buttons

Toolbar button	Button name	Description
	Select	Selects one or more entities. The Select toolset has these buttons: Select Direct; Select All; Modify Selection; Select Inside Polygon; Select Fence; Clear Selection; Marquee; Irregular Marquee; and Clear Marquee.
	Zoom	Gets a closer view of a portion of your drawing. The Zoom toolset has these buttons: Redraw; Zoom Realtime; Zoom In; Zoom Out; Pan Realtime; Pan; Last View; Next View; View Selection; View Page; View Extent; View Save; View Recall.

All-In-One Toolbar buttons (*continued*)

Toolbar button	Button name	Description
	Line	Draws lines and line variations. The Line toolset has these buttons: Line Single; Line Segment; Line Multiple; Line Double; Line Tangent; Line Perpendicular; Line Angle.
	Arc	Draws circular arcs and elliptical arcs. The Arc toolset has these buttons: 3 Point Arc; 2 Points and Center Arc; 2 Points and Angle Arc; Elliptical Arc Rectangle; Elliptical Arc Axes.
	Circle	Draws circles and ellipses. The Circle toolset has these buttons: Center, Side Circle; Side, Side Circle; 3 Point Circle; Center, Radius Circle; Circle Tangent 2; Circle Tangent 3; Ellipse Rectangle; Ellipse Axes.
	Polyline	Draws polylines. The Polyline toolset has these buttons: Single Polyline; Polyline Segment; Perpendicular Polyline; Center Polyline; Sketch Polyline.
	Polygon	Draws polygons. The Polygon toolset has these buttons: Rectangle; Rotated Rectangle; Regular: Center, Edge; Regular: Edge, Opposite; Regular: Edge, Adjacent; Regular: Center, Radius; Irregular Polygon; Irregular Cloud.
	Curve	Draws fitted and spline curves. The Curve toolset has these buttons: Fitted Curve; Spline Curve.
	Marker	Marks points in your drawing. The Marker toolset has these buttons: Marker Point; Marker Align Entity; Marker Align Endpoint.
	Text	Enters a line or paragraph of text in your drawing.The Text toolset has these buttons: Text Point; Text Rectangle.
	Dimension	Draws dimension lines. The Dimension toolset has these buttons: Horizontal Dimension; Vertical Dimension; Rotated Dimension; Aligned Dimension; Angular Dimension; Radius Dimension; Diameter Dimension; Centerline Dimension; Ordinate Dimension; Leader.
	Duplicate	Creates a duplicate of a selected entity and places it at a specific offset distance. The Duplicate toolset has these buttons: Parallel; Offset.

All-In-One Toolbar buttons (*continued*)

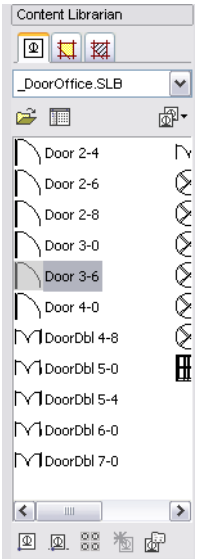
Toolbar button	Button name	Description
	Fill	Creates hatches (a repetitive line pattern in an enclosed area defined by a selection set). The Fill toolset has these buttons: Fill Hatch; Fill Solid Color.
	Picture/Detail View	Inserts bitmap pictures or detail views into the drawing. The Picture/Detail View toolset has these buttons: Picture From File; Detail View.
	Symbol	Manages, creates, places, and duplicates symbols in a drawing. The Symbol toolset has these buttons: Symbol Point; Insert Symbol; Symbol Array; Create Symbol.
	Inquire	Displays information about a drawing and its entities. The Inquire toolset has these buttons: Inquire Entity; Inquire Symbol Count; Inquire Selection; Inquire Drawing; Inquire Coordinate; Inquire Distance; Inquire Angle; Inquire Area.
	Transform	Moves, scales, rubber-stamps, or rotates entities. The Transform toolset has these buttons: Rubber Stamp; Rubber Stamp Array; Translate; Scale; Rotate; Align; Mirror; Stretch; Rectangular Array; Circular Array.
	Trim	Edits the geometry of entities. The Trim toolset has these buttons: Trim Corner; Trim Round; Trim Bevel; Trim Edge; Trim Break; Trim Channel; Trim Divide; Trim Subdivide; Trim Join; Trim Alcove; Trim Union; Trim Intersection; Trim Difference
	Snap	Snaps to a point on the grid. The Snap toolset has these buttons: Snap Off; Gridpoint Snap; Endpoint Snap; Jump Snap; Midpoint Snap; Nearest Snap; Basepoint Snap; Perpendicular Snap; Intersection Snap; 2 Point Intersection; Centerpoint Snap; Tangent Snap; Quadrant Snap; Absolute Input; Relative Input; Set Last (Working) Point.
	Lock	Turns lock modification on and off. The Lock toolset has these buttons: Unlock; Lock X; Lock Y; Ortho Lock; Normal Lock.

Communication Center

Displays product announcements, product support information, and articles and tips of interest. You can learn how to use Communication Center in “Use the Communication Center” on page 104.

Content Librarian

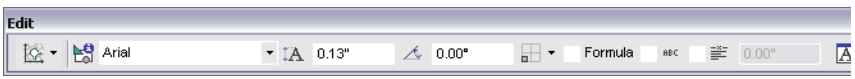
Contains symbols, fill colors, and hatches that you can insert into a drawing. If a wizard was used to create a drawing, the Content Librarian provides symbol libraries specific to that wizard.



The Content Librarian with the _DoorOffice symbol library displayed

Edit Bar

When active, allows you to edit geometric properties of an entity. The function of the edit bar changes depending on the task you are doing. For example, if you select text in the drawing, the controls on the edit bar allow you to edit the text, font, height of the text, and so on.



The edit bar when text is selected in a drawing

Menu Bar

Displays a list of menus and their options. You can also use toolbars and shortcut keys on the keyboard (CTRL+<letter>) to perform the same tasks.

The menu bar

Property Bar

Sets the current layer, color, style, width, and pattern. Any change you make on the property bar affects entities that are currently selected, and those that you draw in the future.

The property bar

To change a setting on the property bar, click the small arrow to display the list of properties, and make a new selection. To apply a new setting to an entity, select the entity you want to change, and then click the property setting on the property bar.

Status Bar

Displays a message area on the left side and the coordinates dials on the right. The message area displays prompts, messages, and step-by-step instructions for most procedures.

The status bar

Two *dials* occupy the right side of the status bar. The Absolute Coordinates dial (on the left side) displays the absolute location of the point (its position in relation to the drawing origin). The Relative Coordinates dial (on the right side) displays the relative location of the point (its position in relation to the last point entered).

Title Bar

Displays the name of the program and the name of the current drawing file. The AutoSketch title bar extends across the top of the application window.

The title bar

Toolbars

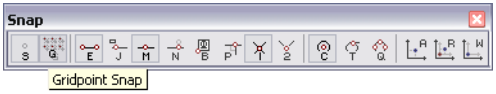
Provide buttons that allow you to do drawing tasks. (You can also use menus to perform the same tasks.) When a toolbar button has a triangle in its lower-right corner, you can click and hold the button to access additional items, called *toolsets*.

The All-In-One toolbar with the Circle toolset displayed

You can move a toolbar by clicking near its left edge and dragging it to its new location. You can also place toolbars next to one another and dock them in the drawing window.

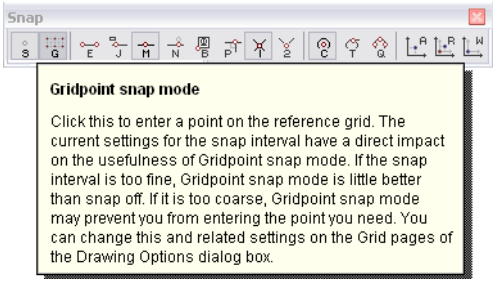
Tooltips

Display the name or the function of toolbar buttons. Hold your pointer over a tool to display its tooltip.



Example of a tooltip

To display a more detailed explanation of the tool, click the Help button (on the Standard toolbar), and then click a toolbar button.



Example of detailed Help for a tool

Basic Features and Functionality

Before you use AutoSketch, there are important features and functionality that you should understand. Understanding the concepts in the following sections is the key to a successful experience of doing this book's exercises. It is strongly recommended that you read this material carefully. Each concept is briefly defined here and described in detail afterward.

- ***Entity.*** A single object, such as a line, polygon, or symbol.
- ***Properties.*** An item of information assigned to an entity. Properties include geometry, layer, pen, pattern, and so on.
- ***Scale.*** The ratio between the size of an entity in its scaled output and the size of the real-world object it represents. For example, if an entity that is 1/4 inch long in its scaled output represents a real-world object that is 1 foot long, the drawing scale is 1/4"=1'0".
- ***Coordinates.*** A pair of numbers that together specify the location of a point in your drawing.
- ***Drawing Origin.*** The point that serves as a location reference for all entities in the drawing. The x- and y-axes cross at the drawing origin. The coordinates of the drawing origin are 0,0.
- ***Reference Grid.*** An on-screen drawing aid consisting of a snap grid and a pattern of lines, crosses, or dots that visually represent the grid.
- ***Grid Origin.*** The point from which the axes of the reference grid extend outward.
- ***Snap.*** A means of entering points using the mouse or keyboard. You can change the snap at any time during most Draw and Edit operations by typing the appropriate keyboard shortcut.
- ***Lock Modifiers.*** Four modifications that you can apply to a snap. Lock modifiers align input with the last point and are applied after the snap.

For detailed information about these concepts, see the Help system.

Entities

Entities are the fundamental elements of a drawing. They can be simple (*base* entities), such as a single line, arc, circle, or polyline, or they can be groups of drawing elements (*compound* entities), such as symbols and dimensions. Entities can also be other elements in your drawing, such as pictures or elements from other drawings. Most entities can be edited. You can resize them and change their properties.

The following table lists the type of entities that you can create in AutoSketch.

Entity	Description
Arc	A portion of a circle. You can use an arc to show a rounded wall, the direction a door swings, and so on.
Circle	A curved line with every point equally distant from the center. You can use a circle to represent a hole, a round object, and so on.
Curve	A polyline that is rendered onscreen and on printed output in a special way. Use curves to create free-form shapes such as curved sidewalks and car fenders.
Detail view	A rectangular area that displays another portion of the drawing defined by a previously saved view.
Dimension	A predefined collection of lines, arcs, markers, and text that display a measurement in the drawing. The text label is updated automatically when you stretch or reshape the dimension.
Ellipse	A closed symmetrical curve that resembles a flattened circle.
Fill	A hidden-line polygon that conforms to the shape of a bounded area and displays either a solid color, a hatch, or a bitmap fill.
Group	A compound entity consisting of individual symbols and entities that AutoSketch treats as a single entity.
Line	An entity that connects two points. You can use a line to represent any straight object such as a water pipe, a wall edge, an electrical connection, or a street.
Marker	An entity that notes a specific point in a drawing.

Entity	Description
OLE Object	An entity that is created in one application and embedded in another. When you double-click a linked OLE object, Windows opens the source application that created it and loads the associated file.
Picture	A picture or bitmap that can be imported and placed in the drawing. AutoSketch treats the raster image like most other entities, allowing you to move, scale, or duplicate the image as needed.
Polygon	A closed polyline that can contain a fill pattern. Use a polygon when you need to know the area of an enclosed region or when you need to fill an area with a solid color, a hatch, or a bitmap fill.
Polyline	A multi-segmented line that AutoSketch treats as a single entity. When a polyline is closed, it becomes a polygon. Use a polyline when you need to know the total length of a series of connected segments.
Symbol	A group of entities that AutoSketch treats as a single entity. Symbols can be stored in libraries for use in multiple drawings.
Text	A text entity that can be any size, can use any TrueType font, and can be rotated at any angle.

Properties

Properties are the individual qualities that define an entity. They are divided into three categories:

- ***Geometric properties*** define an entity's size, position, and so on. AutoSketch assigns geometric properties automatically as you draw and edit.
- ***Graphic properties*** specify the appearance of an entity. Graphic properties include layer, color, width, style, and pattern. AutoSketch assigns these properties as you draw, based on the current settings on the property bar.
- ***Fields*** customize an entity in ways that you define. Define a field by specifying its name, type, and width or precision. A desk symbol, for example, can have fields for model, size, color, and style.

Scale

Drawing scale is the ratio between the actual size of the entities in a drawing and their size on printed output. In conventional drafting, you scale the components of a drawing by using an architectural or engineering scale. In AutoSketch, you simply enter the actual (world) size of an entity, and the software keeps track of the scale for you.

You can create 1:1 drawings in AutoSketch without regard for scale. Specifying a drawing scale, however, has two important benefits. It allows AutoSketch to accurately depict how your drawing will look on a printed page. It also allows you to specify entities such as text, markers, and dimensions by output size.

Any output you plan to measure with an architectural or engineering scale must be printed to scale. When you create scaled output, you can print the entire drawing or a portion of the drawing. The scale used when printing is the current drawing scale.

Coordinates

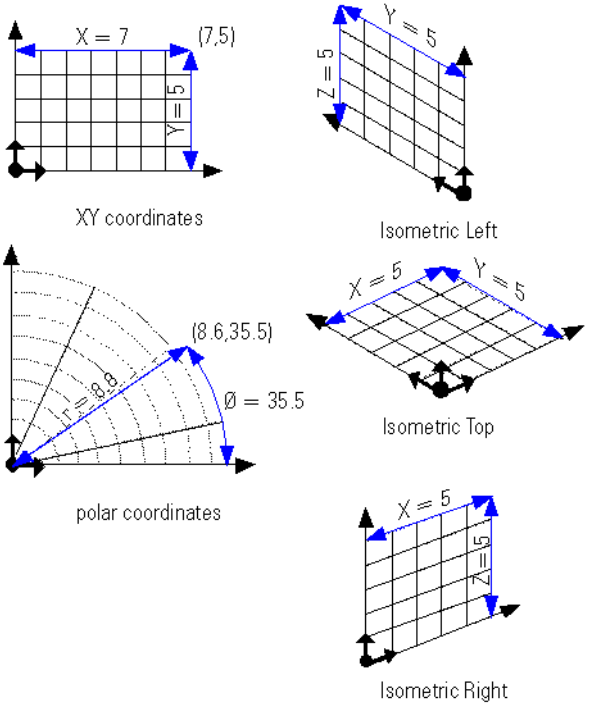
Coordinates are numbers that specify the location of one point in relation to another. This relationship is classified as either absolute or relative.

Absolute coordinates reference the origin of whatever coordinate system is currently being used (for example, the drawing origin or the grid origin).

Relative coordinates reference the last point you entered. They are useful when you want to draw or place another entity a known distance from another entity or point.

AutoSketch expresses location in three ways: xy (Cartesian), polar, and isometric coordinates. X- and y-coordinates express location in terms of horizontal and vertical distances from another point. Polar coordinates express location in terms of distance (radius) and angle. For example, the xy coordinates 7,5 are equivalent to the polar coordinates 8.6,35.5.

Isometric coordinates add a third axis (z) to the expression. Isometric drawings are often used to create two-dimensional views of a three-dimensional object.

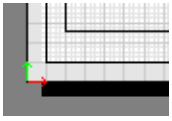


Examples of coordinates

Drawing Origin

The drawing origin displays the *x* (horizontal), *y* (vertical), and (if isometric) *z* coordinates of a drawing. AutoSketch locates most points in relation to the drawing origin.

The drawing origin is shown on screen as colored arrows.

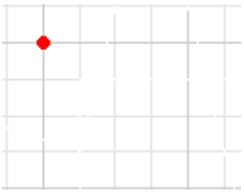


Example of the drawing origin arrows in the lower-left corner of a drawing

If you move the drawing origin, the entire drawing shifts to reflect that change. The drawing origin does not appear when you print the drawing. Normally, the drawing origin is located at the lower-left corner of a drawing. If you need to move it, you can center the drawing origin or relocate it.

Reference Grid

A reference grid is a visual drawing aid that contains a pattern of horizontal and vertical lines or dots that represent a grid. Use gridpoint snaps to make your drawing precise.



Example of a first point snapping to a grid

There are three types of reference grids available in AutoSketch, each suited for different purposes.

- The default grid is *rectangular*, with snap intervals and lines that parallel the x- and y-axes. This grid is the standard reference tool for most two-dimensional drawings.
- *Circular* grids extend radially from the grid origin. They provide an excellent reference tool for drawings that require alignment of points along an arc or circle, such as a mechanical drawing of a gear.
- *Isometric* grids align along three major axes, instead of two. This allows you to create two-dimensional drawings of three-dimensional objects.

You can modify the settings for each of the reference grids.

The following table lists the Grid tools you can use in AutoSketch. These buttons are located on the Grid toolbar.

Toolbar button	Button name	Description
	Rectangular	The most commonly used, is useful for most two-dimensional drawings.
	Circular	Aligns grid lines along an arc or circle. The radial grid lines allow you to enter such points precisely. When you set up a circular grid, you may need to reposition the grid origin so that the radial lines of the grid are aligned correctly on the page.
	Isometric Top	Aligns snap and grid lines along 30- and 150-degree axes.

Toolbar button	Button name	Description
	Isometric Left	Aligns snap and grid lines along 90- and 150-degree axes.
	Isometric Right	Aligns snap and grid lines along 90- and 30-degree axes.
	Double Grid Size	Doubles the current grid size.
	Halve Grid Size	Decreases the current grid size by half.

Tip You can also change settings using the Edit Grid button on the Standard toolbar.

Grid Origin

The grid origin is similar to the drawing origin in function and appearance. However, the grid origin serves as a reference point for grid coordinates only. By default, the grid origin is located at the drawing coordinates 0,0. You can move the grid origin of rectangular, circular, or isometric reference grids.

Snap

Using snap, you can draw with real precision by identifying exact points such as an entity's midpoint, endpoint, or centerpoint. These points are called snap points because when you click near one, the point is snapped to the exact point shown.

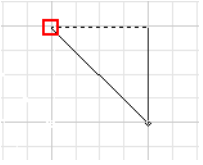
There are 15 ways to snap to a point in AutoSketch. These correspond with the 15 snaps you can choose by clicking their buttons on the All-In-One toolbar, Snap toolbar, or by typing the letter shown on the button.

When snaps are active, a red AutoPoint Indicator is displayed on the grid. As you move the pointer over a drawing, each type of snap point displays a different symbol. (You may notice these snap types when you create simple entities in the first tutorial.) The following table lists the default snap types.

Symbol	Snap Type	Description
	Gridpoint snap	Snaps to the reference grid.
	Endpoint snap	Snaps to the endpoint of an entity.
	Midpoint snap	Snaps to the midpoint of a line, polyline segment, etc.
	Intersection snap	Snaps to intersection points.
	Centerpoint snap	Snaps to the center of an arc, circle, polygon, or bulged poly-segment.

Lock Modifier

You can align input with the last point by using a lock modifier. If a lock modifier is active, a dotted line extends from the AutoPoint Indicator to the actual point, as constrained by the lock modifier. For example, if you draw a diagonal line from top to bottom, and then activate Endpoint snap and the Y-axis lock modifier, the square AutoPoint Indicator identifies the endpoint nearest the pointer, but a dotted line extends to the potential snap point based on the current snap and lock modifier.



Example of a line drawn with endpoint snap and y-axis lock modifier turned on

There are four automatic modifications you can have AutoSketch make to the point you enter. These lock modifiers force the point you enter into horizontal, vertical, orthogonal, or “normal” alignment with the last point. At any time in the drawing or editing process, you can apply a lock modifier by clicking its button on the All-In-One toolbar or by typing the letter shown on the button.

Tutorial 1 — Create and Trim Entities

In this tutorial, you learn how to start AutoSketch®, create a drawing, and create entities. You also learn how to trim entities using several methods.

More information about each of the concepts in this tutorial is available in the Help system.

In this tutorial

- Start AutoSketch
- Create Simple Entities
- Create Lines
- Create Other Simple Entities
- Trim an Entity

Start AutoSketch

In this exercise, you learn to

- ❒ Start AutoSketch.
- ❒ Create a drawing from scratch.

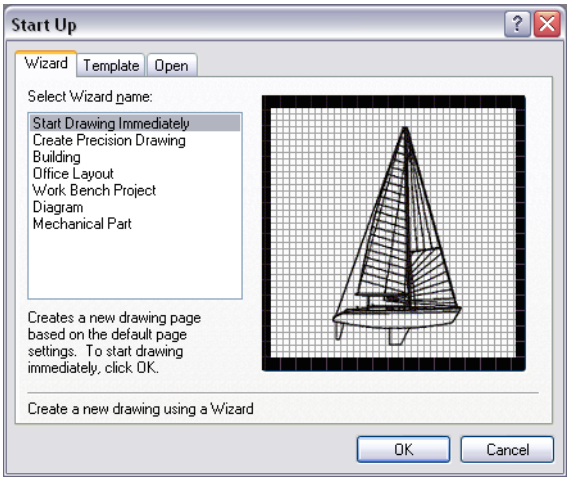
Note At the end of each exercise, you can take a break or move to the next exercise. Be sure to save your work at the end of each exercise, because each subsequent exercise builds on the one before it.

To start AutoSketch

1. On the Start menu (Windows), click All Programs (or Programs) ➤ Autodesk AutoSketch 9.
2. In the Tip of the Day dialog box, read the tip and click Close.

To start a drawing

1. In the Start Up dialog box, Wizard tab, select Start Drawing Immediately.



2. Click OK.

Create Simple Entities

In the exercises that follow, you become familiar with simple entities by creating lines, arcs, circles, polygons, and polylines. While you create entities, you also become familiar with the AutoSketch drawing window and how the menus and toolbars can be used to create entities. In a later tutorial, you learn how to create useful drawings with entities and symbols. But for now, have fun creating simple entities.

Create Lines

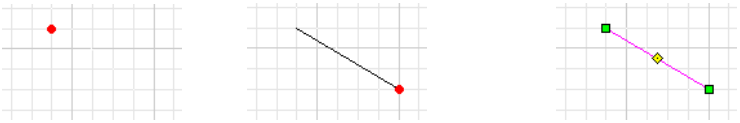
In this exercise, you learn to

- ❒ Create a single line, a multiline, and a double line.
- ❒ Create lines using the Draw menu and the All-In-One toolbar.
- ❒ Right-click a toolbar button to access related buttons on a toolset.
- ❒ Right-click a command to end it.
- ❒ Use the SHIFT key to select multiple entities.
- ❒ Delete entities.

A line is an entity that connects two points. You can use a line to represent any straight object.

To create a line

1. In AutoSketch, on the File menu, click New.
2. In the New dialog box, click Start Drawing Immediately, and then click OK.
3. On the Draw menu, click Line ➤ Single.
4. In the drawing window, click anywhere to create the startpoint of the line. Then, click another point in the drawing window to create the endpoint. Right-click to end the command.

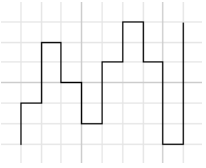


Example of a line produced with the Line Single option

You have just created a line, your first computer-drawn entity. Continue to draw lines until you feel comfortable with the action. When you are ready, you can create a line that has multiple points.

5 On the Draw menu, click Line ➤ Multiple.

6 In the drawing window, click anywhere to create the startpoint of the line. Then, click another point in the drawing window to create a second line point. Continue clicking to create additional points. When you are finished, right-click to complete the line.



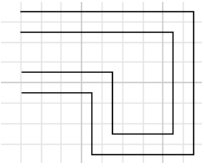
Example of a line produced with the Line Multiple option

Now, use the All-In-One toolbar to create a line. When you click and hold the Line button on the All-In-One toolbar, you see additional buttons called *toolsets*. Hovering over a button displays a tooltip with the name of the button. When you end the command, the new button is shown on the toolbar. The last button used is what appears on the toolbar, until a new button is used.

7 On the All-In-One toolbar, click and hold the Line button.

8 On the toolset, drag the pointer until you locate Line Double, and then release the mouse button.

9 Click anywhere in the drawing to create the startpoint of the line. Then, click another point in the drawing window to create a second line point. Continue clicking to create additional points. When you are finished, right-click to complete the line.



Example of a line produced with the Line Double option

You may want to clean up your drawing at this point so that you have room to draw more entities.

To delete entities

- On the keyboard, press CTRL+A to select all entities in the drawing, and then press the DELETE key.

Now that you understand how to create different kinds of lines using the Draw menu and the All-In-One toolbar, you can create other entities.

Create Other Simple Entities

In this exercise, you learn to

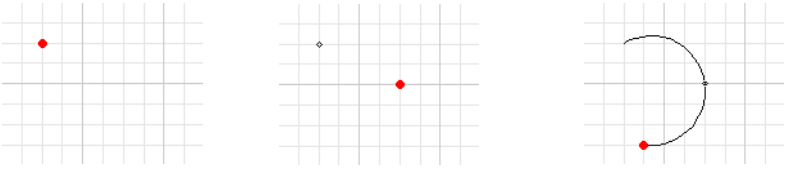
- ❒ Create arcs, circles, polylines, and polygons.
- ❒ Select startpoints, midpoints, and endpoints.
- ❒ Use the All-In-One toolbar to create entities.
- ❒ Use the All-In-One toolbar to access toolset tools.

In this exercise, you create variations of an arc, a circle, a polyline, and a polygon.

- **Arc.** A segment of a circle defined by a centerpoint, radius, starting angle, and included angle.
- **Circle.** An entity with a centerpoint and a radius.
- **Polyline.** A multi-segmented line (the segments can be straight or curved). Use a polyline to determine the total length of a series of connected sgments.
- **Polygon.** A polyline whose startpoints and endpoints are connected (closed) to create a shape with multiple sides.

To create an arc

1. On the Draw menu, click Arc ➤ 3 Points.
2. In the drawing window, click a point to begin the arc, click a second point, and then click another point to end the arc. Try this a few times to get familiar with it.



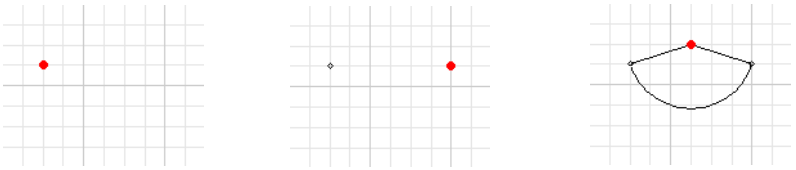
Example of an arc produced with the Arc 3 Points option

Now, use the All-In-One toolbar to create a different kind of arc.

3 On the All-In-One toolbar, click and hold the Arc button.

4 On the toolset, drag the pointer until you locate 2 Points and Center, and then release the mouse button.

5 In the drawing window, click a point to begin the arc. Click another point to mark the second point. Then, click again to mark the centerpoint.



Example of an arc produced with the 2 Points and Center option

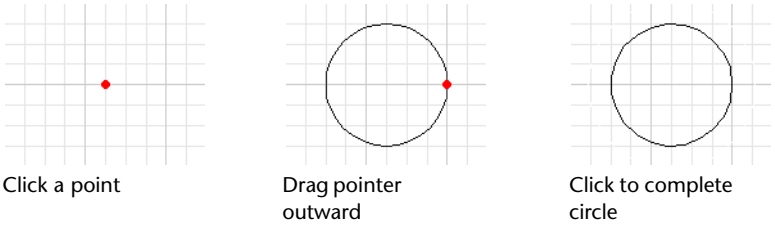
Tip The status bar, in the lower-left corner of the drawing window, displays prompts that describe the next step in a procedure. If you're unsure what the next step is, look at the status bar for prompts.

6 When you finish working with arcs, delete the entities.

To create a circle

1 On the Draw menu, click Circle ➤ Center, Side.

2 In the drawing window, click a point to set the center of the circle, and then drag the pointer out from the center until the circle is the size you want. Then, click at that point to complete the circle. Try this a few times to get familiar with it.



Example of a circle produced with the Center, Side option

3 On the All-In-One toolbar, click and hold the Circle button, and on the toolset, click any of the Circle buttons. Create new circles in your drawing.

4 When you finish working with circles, delete the entities.

To create a polyline

1 On the All-In-One toolbar, click and hold the Polyline button.

2 On the toolset, drag the pointer until you locate Single Polyline, and then release the mouse button.

3 In the drawing window, click a point to begin the polyline, and then click two more points to create a polyline that is shaped like the letter "V". When you have created your polyline, right-click twice to complete the line and end the polyline command. Try this a few times to get familiar with it.

Example of a polyline produced with the Single Polyline option

4 On the All-In-One toolbar, click and hold the Polyline button, and on the toolset, click any of the Polyline buttons. Experiment with the different kinds of polylines you can create.

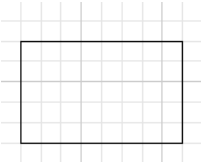
5 When you finish working with polylines, delete the entities.

To create a polygon

1 On the All-In-One toolbar, click the Polygon button.

2 On the toolset, drag the pointer until you locate Rectangle, and then release the mouse button.

3 In the drawing window, click a point to begin the rectangle, and then click another point to end the rectangle.



Example of a polygon produced with the Rectangle option

4 On the All-In-One toolbar, click and hold the Polygon button, and on the toolset, click any of the Polygon buttons. Create new polygons in your drawing.

   When you finish working with polygons, close the drawing.

5 On the File menu, click Close. In the Save Changes to Drawing message, click No.

Trim an Entity

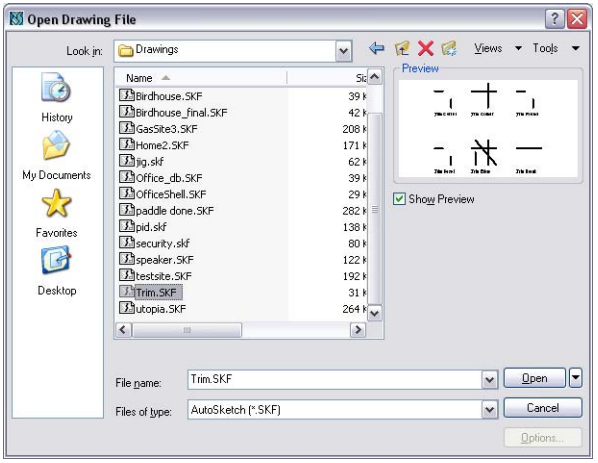
In this exercise, you learn to

- ❒ Shorten and lengthen an entity.
- ❒ Create rounded and beveled corners.
- ❒ Break apart and divide an entity.

Trimming allows you to shorten and lengthen entities so they can meet at a specific point to create rounded and beveled corners, or to break apart and divide entities.

To trim an entity

1 In AutoSketch, on the File menu, click Open.

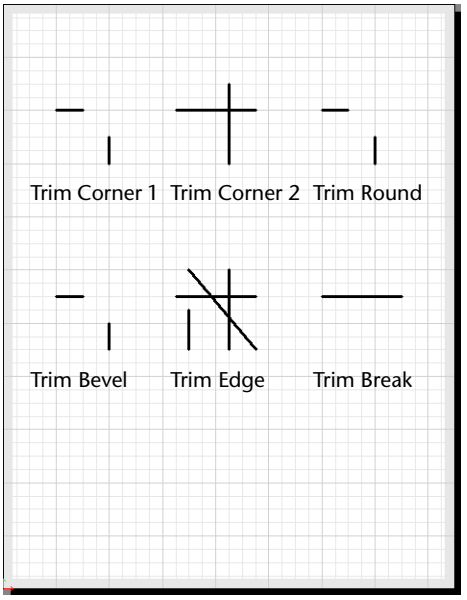


2 In the Open Drawing File dialog box, navigate to the following location:

   C:\Program Files\Autodesk\AutoSketch9\Drawings

3 In the list of files, select *Trim.skf*, and then click Open.

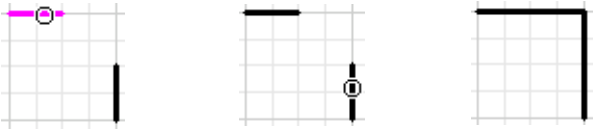
   The following drawing is opened in AutoSketch.



First, *join* two perpendicular lines to create an intersection.

4 On the All-In-One toolbar, click Trim Corner.

5 In the Trim Corner 1 section of the drawing, click the horizontal line. Then, click the vertical line to the right of that horizontal line. Right-click to end the command.

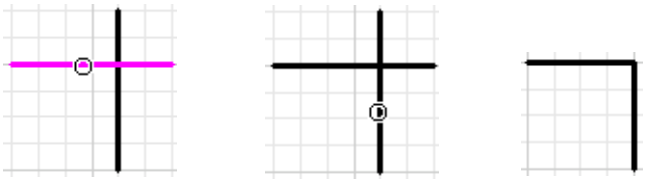


Example of two perpendicular lines joining to form a corner

Next, use the same Trim Corner button to *remove* excess lines and form a corner.

6 On the All-In-One toolbar, click Trim Corner.

7 In the Trim Corner 2 section of the drawing, click the left side of the horizontal line. Then, click the lower part of the vertical line. Right-click to end the command.



Example of two intersecting lines trimmed to form a corner

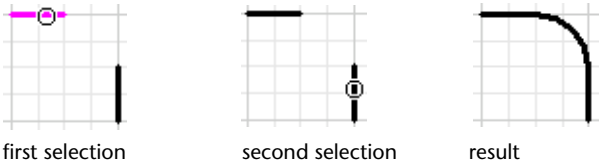
Now, create a rounded corner.

8 On the All-In-One toolbar, click and hold the Trim Corner button.

9 On the toolset, drag the pointer to select Trim Round.

10 In the drawing, locate Trim Round, in the upper-right section of the drawing.

11 In the Trim Round section of the drawing, click the horizontal line. Then, click the vertical line to the right of that horizontal line. Right-click to end the command.



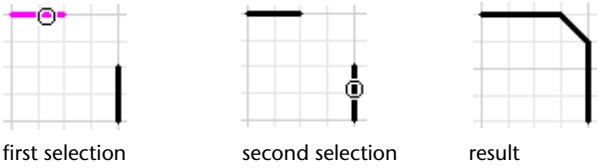
Example of two perpendicular lines joining to form a rounded corner

Next, create a beveled corner.

12 On the All-In-One toolbar, click and hold the Trim Round button.

13 On the toolset, drag the pointer to select Trim Bevel.

14 In the Trim Bevel section of the drawing, click the horizontal line. Then, click the vertical line to the right of that horizontal line. Right-click to end the command.



Example of two perpendicular lines joining to form a beveled corner

Next, join two divided lines and then trim the extraneous lines in an intersection.

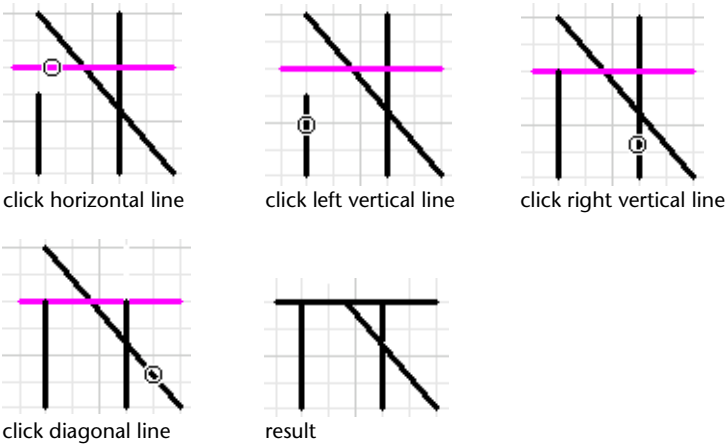
15 On the All-In-One toolbar, click and hold the Trim Bevel button.

16 On the toolset, drag the pointer to select Trim Edge.

17 In the Trim Edge section of the drawing, do the following, in order:

- Click the left part of the horizontal line.
- Click the left vertical line.
- Click the right vertical line, just below the diagonal line.
- Click the diagonal line just to the right of the vertical line that you just selected.

Your drawing should match the last picture in the following sequence of illustrations.



Example of two divided perpendicular lines joined and extraneous lines trimmed

Next, create a break in a line.

18 On the All-In-One toolbar, click and hold the Trim Edge button.

19 On the toolset, drag the pointer to select Trim Break.

20 In the Trim Break section of the drawing, click anywhere on the horizontal line. Then, move the pointer to the center of the line, and click to create the break point. Right-click to end the command.

Example of a line broken into two equal sections

21 On the File menu, click Close.

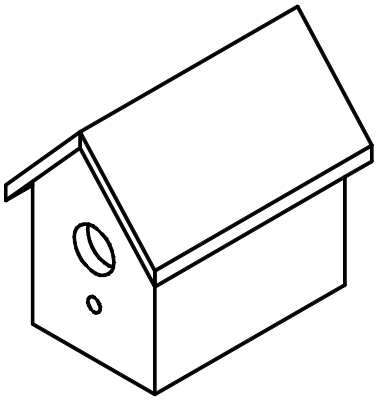
22 In the Save Changes to Drawing message, click No.

Now that you understand how to create different kinds of entities using the Draw menu and the All-In-One toolbar, and how to trim entities, you can move on to more challenging exercises.

Tutorial 2 — Create a Birdhouse Drawing

In this tutorial, you learn how to use AutoSketch® to create a birdhouse drawing. You create a single entity from scratch, and you create an entity by grouping and rubber-stamping an existing entity. You also add dimensions and a title, and then place them on the appropriate layers. Finally, you print the drawing.

More information about each of the concepts in this tutorial is available in the Help system.



In this tutorial

Introduction

Before you start drawing, make sure the drawing setup works for your needs. Consider the page size, page layout, scale, grid, layers, and so on.

Once you set up your drawing, you can begin to draw entities, move them to fit the page, set dimensions, and enter annotations. By the time you have completed this exercise, you will have created a drawing that you can use to build an actual birdhouse.

Note At the end of each exercise, you can take a break or move to the next exercise. Be sure to save your work at the end of each exercise, because each subsequent exercise builds on the one before it.

Set Up the Birdhouse Drawing

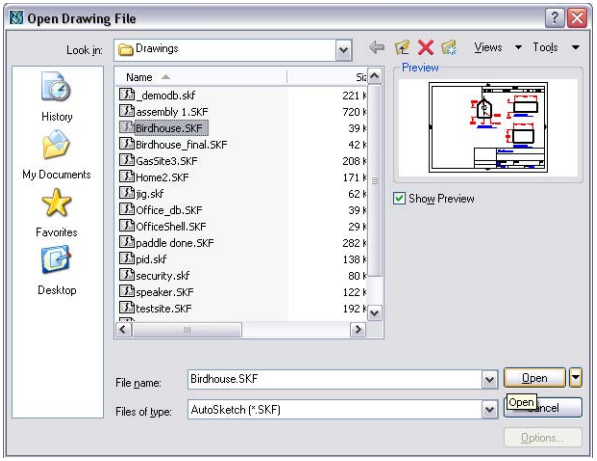
Before you create the birdhouse, make sure your drawing is properly set up. You can set up the Birdhouse drawing by resetting the interface to its original state.

In this exercise, you learn to

- ❒ Open a drawing.
- ❒ Reset the interface.

To set up the Birdhouse drawing

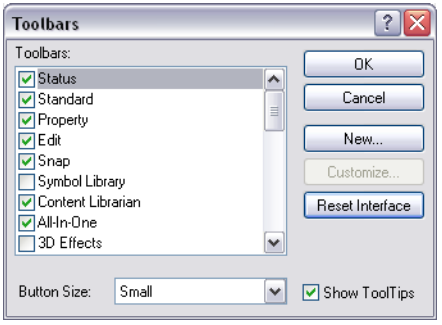
1. On the File menu, click Open.
2. In the Open dialog box, navigate to the following location:
   C:\Program Files\Autodesk\AutoSketch9\Drawings



In the *Drawings* folder, select *Birdhouse.skf*, and then click Open.

Now, reset the interface to make sure your settings match the tutorial's instructions.

3 In AutoSketch, on the View menu, click Toolbars.



4 In the Toolbars dialog box, in the lower-right corner, click Reset Interface.

5 In the warning message that is displayed, click Yes.

Your drawing window is now opened and set up for this tutorial.

Create the Floor of the Birdhouse

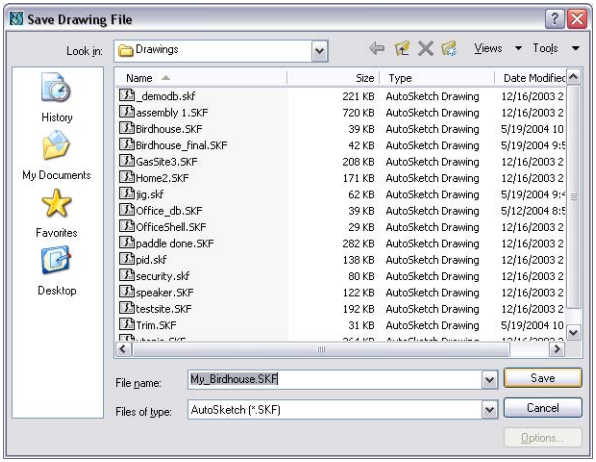
In this exercise, you learn to

- ❒ Rename and save the drawing file.
- ❒ Zoom in to a section of a drawing.
- ❒ Use the Absolute Coordinates dial to place an entity precisely.
- ❒ Set line widths.

In the drawing for this tutorial, entities already exist for the back, roof, and sides of the birdhouse. In this exercise, save the drawing file, and then create a rectangular entity that represents the floor of the birdhouse.

To create the birdhouse floor

1 In AutoSketch, on the File menu, click Save As.

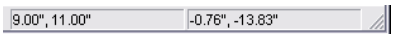


2 In the Save Drawing File dialog box, in the File Name box, type **My_Birdhouse**, and then click Save.

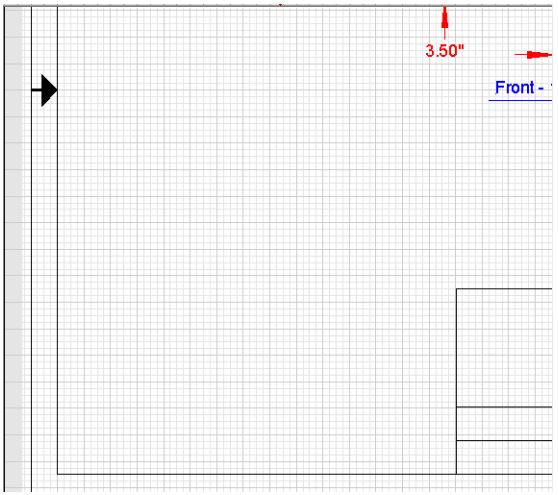
3 On the Standard toolbar, click the Zoom In button.

Now, use absolute coordinates to zoom in to a precise location in the drawing. The Absolute Coordinates dial, the first set of coordinates displayed on the right side of the status bar (at the bottom of the AutoSketch window), places your points in exact locations.

4 In the drawing window, move the pointer, and click when 9.00", 11.00" is displayed on the status bar in the Absolute Coordinates dial.



You should be zoomed in to the lower-left corner of the drawing, as shown in the following illustration.

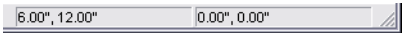


5 On the All-In-One toolbar, click and hold the Polygon button.

6 On the toolset, drag the pointer to select Rectangle.

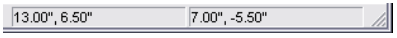
First, create the floor of the birdhouse. The floor measures 7 inches wide by 5.5 inches long.

7 Move the pointer. Click when 6.00", 12.00" is displayed on the status bar in the Absolute Coordinates dial.



You have just set the first point. From the point of origin, the first point is located 6 inches along the x axis and 12 inches along the y axis.

8 Drag your pointer to the right and down until the Absolute Coordinates dial displays the second point at 13.00", 6.50". Click to place the second point. Then, right-click to end the command.

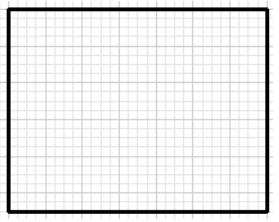


Now, set the line width of the rectangle to match the line widths of the other entities in this drawing (0.039").

9 Click the rectangle you just created.

10 On the property bar, locate the Width button. Click the arrow to the right of the button, and from the list of line widths, select 0.039".

The rectangle should look similar to the following illustration:



11 On the File menu, click Save. Do not close the drawing.

Add Dimensions to the Floor of the Birdhouse

In this exercise, you learn to

- ❒ Add horizontal and vertical dimensions.
- ❒ Move a dimension closer to its entity.
- ❒ Place dimensions on the Dimensions layer.

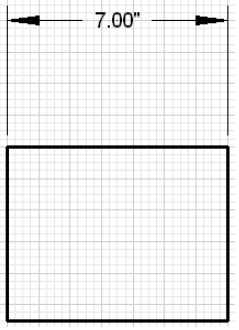
You have just created the first entity for the birdhouse. Next, add dimensions to the drawing.

To add dimensions to the birdhouse floor

1 On the All-In-One toolbar, click and hold the Dimension button.

2 On the toolset, drag the pointer to select Horizontal Dimension.

3 Click the upper-left corner of the floor (the rectangle you just created), and click again when the pointer snaps to the upper-right corner. Then, right-click to end the Dimension command.

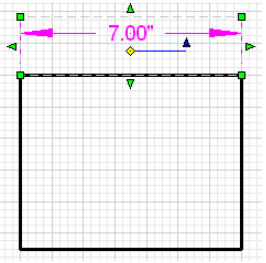


You have just created the horizontal dimension. Now, move the dimension closer to the rectangle.

4 Click the dimension you just created.

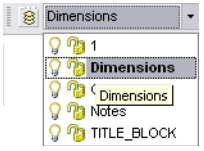
5 Click and hold the green triangle at the top of the selected dimension, and drag the pointer down until the dimension is closer to the rectangle. Then, release the mouse button.

The dimension placement should look similar to the following illustration:



The dimension should still be selected (you did not right-click to end the command). You can now place the dimension on the Dimensions layer.

6 On the property bar, locate the Layer button. Click the arrow to the right of the current layer, and then select Dimensions.



7 Click to the right of the dimension to deselect it.

Notice that the dimension is now red, the same color as the other dimensions in this drawing.

Note Now that the Dimensions layer is active, any entities you create are placed on the Dimensions layer until you change the layer. You learn more about how to change layers later.

Next, create the vertical dimension.

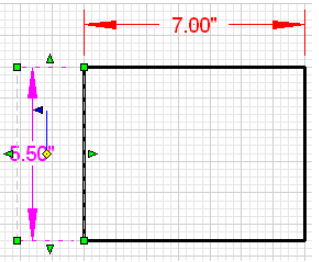
8 On the All-In-One toolbar, click and hold the Horizontal Dimension button.

9 On the toolset, drag the pointer to select Vertical Dimension.

10 Click the lower-left corner of the floor, and drag the pointer up until you snap to the upper-left corner. Click to set the vertical dimension. Then, right-click to end the command.

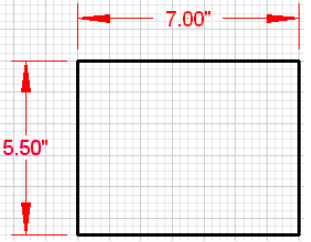
11 Click the dimension you just created. Then, click and hold the arrow to the left of the dimension, and drag the pointer to the right until you can place the dimension closer to the line you just dimensioned.

Your drawing should look similar to the following illustration.



12 Release the mouse button.

The dimension location should look similar to the following illustration:



Because you selected the Dimensions layer in step 6, the vertical dimension you just created is automatically added to the Dimensions layer.

13 Save your work.

Add a Title to the Floor of the Birdhouse

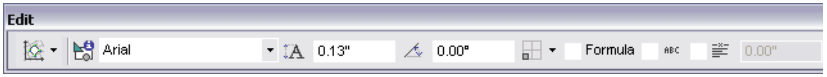
In this exercise, you learn to

- ❒ Add a title to an entity.
- ❒ Adjust the text height using the edit bar.
- ❒ Add a line below the title.
- ❒ Adjust the line width using the property bar.
- ❒ Place the title on the Notes layer.

When you add a title to an entity, use the edit bar.

Add a title and place it on the Notes layer

1. On the All-In-One toolbar, click and hold the Text button.
2. On the toolset, drag the pointer to select Text Point.
3. On the edit bar, in the Text Height box, enter **.13**.



Note The edit bar is dynamic, which means that the controls displayed on it adjust to the current command.

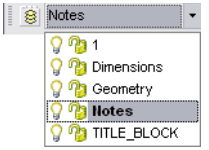
4. Just below the floor entity, click to place the text at the absolute coordinates 6.00", 5.00", and then type **Bottom - 1 required**.

5. On the property bar, locate the Width button. Click the arrow to the right of the button, and from the list of line widths, select Hairline.

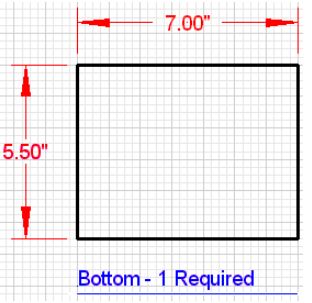
   Now, place a line just below the text you created.

6. On the All-In-One toolbar, click and hold the Line button.

7. On the toolset, drag the pointer to select Line Single.
8. Click the first point with the absolute coordinates at 6.00", 4.75". Then, drag the line to the right, and click when the absolute coordinates read 13.00", 4.75". Right-click to end the command.
9. Click the line.
10. With the text line still selected, press the SHIFT key, and then click the text just above the line.
11. On the property bar, click the arrow to the right of the current layer, and then click Notes.



Once the text and text line are added to the Notes layer, they are blue, the same color as the other text in this drawing.



12 Save your work.

Create the Back of the Birdhouse by Grouping and Rubber-Stamping

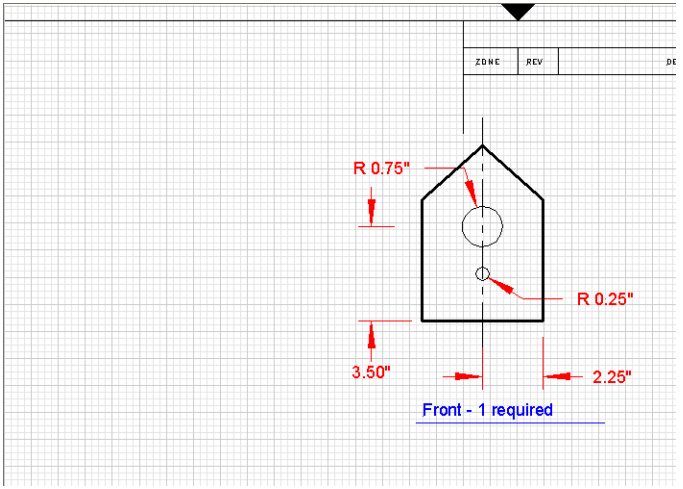
In this exercise, you learn to

- ❒ Pan to a section in the drawing.
- ❒ Select individual entities.
- ❒ Select multiple entities using the SHIFT key.
- ❒ Group an entity.
- ❒ Use the Absolute Coordinates dial to place an entity.
- ❒ Rubber-stamp an entity.
- ❒ Add an entity to the Geometry layer.
- ❒ Explode an entity.

You are ready to create the back of the birdhouse by grouping the front entity and then rubber-stamping the entity. The entity you create in this exercise is the same shape and size as the entity you rubber-stamped, but with slight modifications.

To group the front

1 On the standard toolbar, click Pan Realtime. Pan up and to the left of the drawing until the front of the birdhouse takes up the right half of the drawing window, and the left half contains no entities. Then, right-click and click Cancel to end the Pan command.

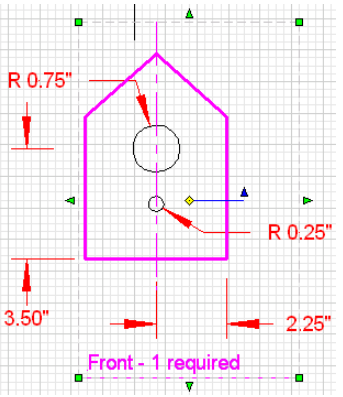


2 On the front of the birdhouse, click the left vertical line, and while pressing the SHIFT key, click the following items:

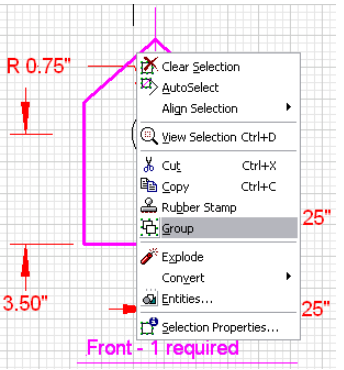
- The remaining four lines that make up the front entity
- The vertical line that runs through the middle of the entity
- The title “Front - 1 Required”
- The line below the title

Note Do not click any dimensions or circles.

The entities you selected are magenta in the drawing, and the entire selected area looks like the following illustration:



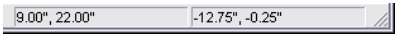
3 Right-click any of the selected entities, and then click Group.



The items you selected are grouped into a new, single entity. You can now duplicate the grouped entity by using a tool called the *rubber stamp*.

4 With the grouped entity still selected, on the All-In-One toolbar, click the Rubber Stamp button.

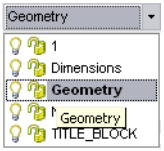
5 Move the pointer to the left of the entity you just selected. Click when 9.00", 22.00" is displayed in the Absolute Coordinates dial, and then right-click to end the Rubber Stamp command.



You have just created a new entity that has the same properties as the original entity. Place the entity on the Geometry layer.

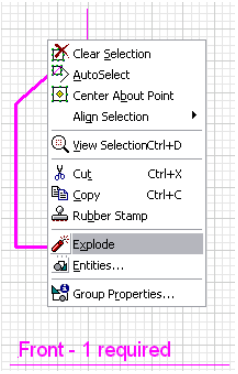
6 Click the grouped entity.

7 On the property bar, click the arrow to the right of the current layer, and then select Geometry.

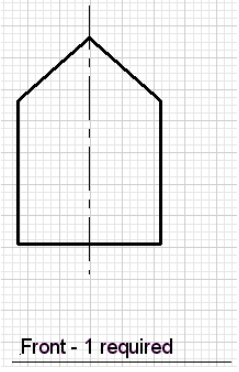


Once the grouped entity is added to the Geometry layer and is no longer selected, it becomes black, the same color as the other entities in this drawing.

8 Click the entity you just added to the Geometry layer. Right-click, and then click Explode.



The selection is now ungrouped, or *exploded*.



9 Save your work.

Add Dimensions to the Back

In this exercise, you learn to

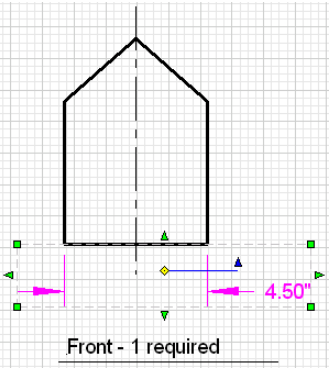
- ❒ Add horizontal and vertical dimensions.
- ❒ Move a dimension closer to its entity.
- ❒ Add an angular dimension.
- ❒ Select multiple entities using the SHIFT key.
- ❒ Place dimensions on the Dimensions layer.

In an earlier exercise in this tutorial, you learned how to create dimensions and add them to the Dimensions layer. Now, in addition to horizontal and vertical dimensions, you create an angular dimension for the birdhouse roof.

To add dimensions to the back

1. On the All-In-One toolbar, click and hold the Dimension button.
2. On the toolset, drag the pointer to select Horizontal Dimension.
3. Click the lower-left corner of the entity.
4. Press and hold the CTRL key, and then move the pointer until it snaps to the lower-right corner.
5. Right-click to end the command.
6. Click the horizontal dimension you just created. Then, click the green downward-pointing arrow at the bottom of the dimension, and drag the dimension up until it is close to the bottom of the entity.

   Your drawing should look similar to the following illustration.



   Next, create the vertical dimension.

7. On the All-In-One toolbar, click and hold the Dimension button.

8 On the toolset, drag the pointer to select Vertical Dimension.

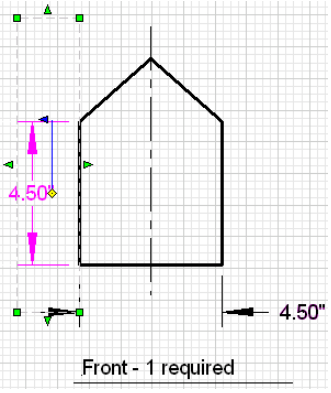
9 Click the lower-left corner of the entity, and drag the pointer up until you snap to the upper-left corner. Click to set the vertical dimension.

You need to move the dimension closer to the entity.

Note You may need to pan to the left a little to see the dimension.

10 Click the vertical dimension you just created. Then, click and hold the arrow to the left of the dimension, and drag the dimension to the right until it is close to the vertical line you just dimensioned. Right-click to end the command.

Your drawing should look similar to the following illustration.



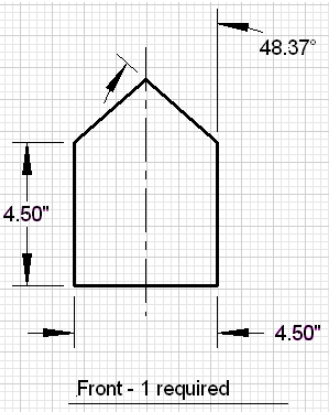
Next, create the angular dimension.

11 On the All-In-One toolbar, click and hold the Dimension button.

12 On the toolset, drag the pointer to select Angular Dimension.

13 Click anywhere on the sloped line on the right to select it. Drag the pointer straight up, and when the Absolute Coordinates dial shows 9.50", 28.00", click to place the dimension.

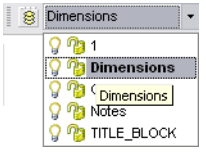
Following is an illustration of the entity after all dimensions are added and moved.



Now, place the dimensions on the Dimensions layer.

14 While holding the SHIFT key, click all three dimensions.

15 On the property bar, locate the Layer button. Click the arrow to the right of the current layer, and then select Dimensions.



16 Click to the right of the entity.

Once the dimensions are added to the Dimensions layer, they are red, the same color as the other dimensions in this drawing.

17 Save your work.

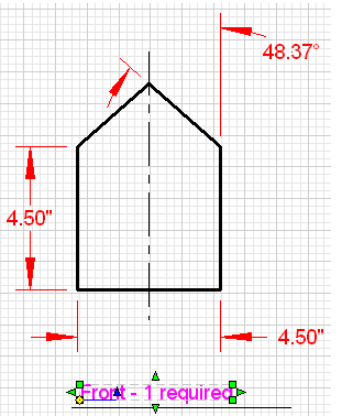
Add a Title to the Back

In this exercise, you learn to

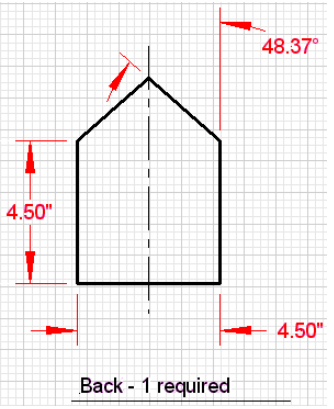
- ❒ Add a title to the back entity by editing an existing title.
- ❒ Add the new title to the Notes layer.

Add a title to the back

1 Below the entity that you just added dimensions to, click the title “Front - 1 Required”. Right-click, and then click Edit Text.



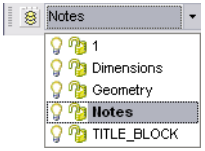
2 In the title, select the word “Front”, and type the word **Back**.



Now, add the new text and the text line to the Notes layer.

3 Click the text line, press the SHIFT key, and then click the text just above the line.

4 On the property bar, click the arrow to the right of the current layer, and then select Notes.



5 Save your work.

You have completed the birdhouse drawing.

Print the Completed Birdhouse Drawing

In this exercise, you learn to

- ❒ Set up the drawing for printing.
- ❒ Print the drawing.

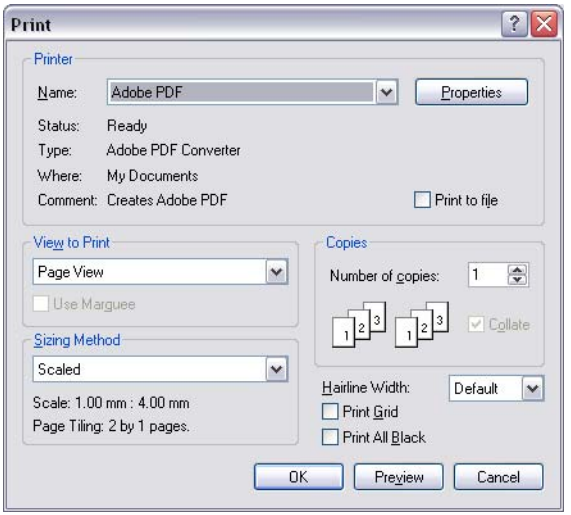
In the final exercise of this tutorial, you can print the completed birdhouse drawing to see all the work that you've done.

To print a drawing

1 On the File menu, click Print.

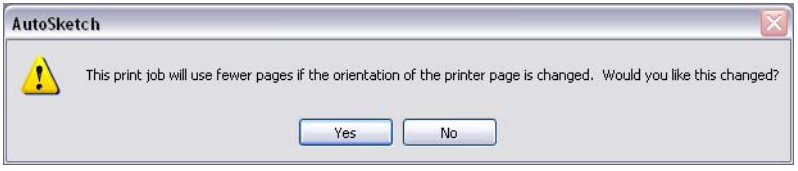
2 In the Print dialog box, change the following settings:

- In the View to Print drop-down list, click Page View.
- In the Sizing Method drop-down list, click Scaled.
- (Optional) To print the grid, click the check box next to Print Grid.



3 Click OK to print the drawing.

4 If the following message is displayed, click Yes to change the orientation of the page, or click No to retain the page orientation.



Congratulations! You have finished the second tutorial, where you learned how to complete and print the birdhouse drawing.

Tutorial 3 — Create an Office Layout Drawing

In this tutorial, you learn how to use the Office Layout wizard to set up a drawing. You also learn how to set units, grid, and scale for the drawing, use symbols to represent the interior components of your drawing, replace symbols, and create a circular array.

More information about each of the concepts in this tutorial is available in the Help system.

In this tutorial

- Start a Drawing Using the Office Layout Wizard
- Set Units, Reference Grid, and Scale
- Create Office Partition Walls
- Add Doors and Windows
- Add Furniture
- Create a Round Table and Chairs

Start a Drawing Using the Office Layout Wizard

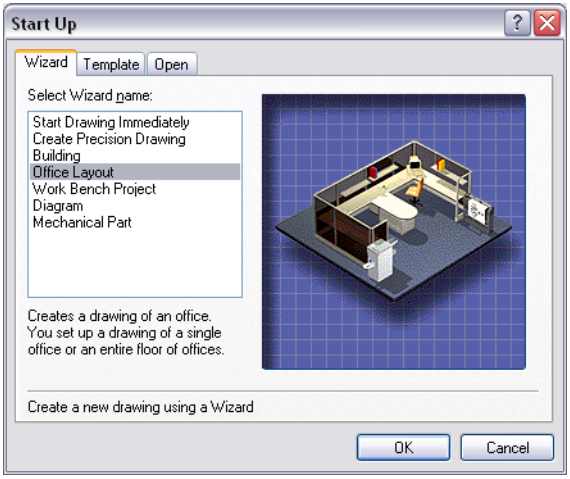
In this exercise, you learn to

❒ Set up an office layout drawing using the Office Layout wizard.

The Office Layout wizard, like the other wizards provided by AutoSketch®, steps you through predefined options to create a drawing that will best suit your project. You can always change the options after you've completed the wizard. This tutorial guides you through using the wizard.

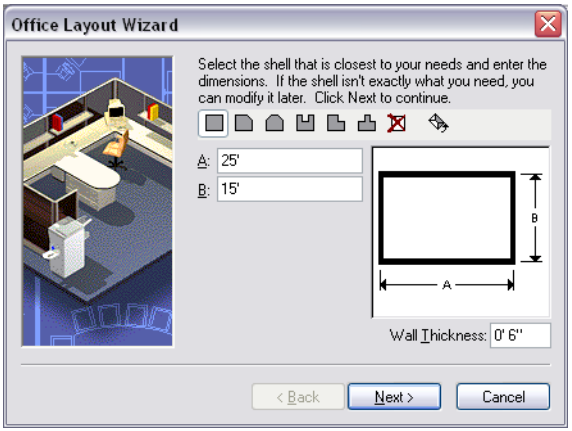
To start a drawing using the Office Layout wizard

1 In the Start Up dialog box, Wizard tab, select Office Layout.



2 Click OK.

On the next page of the wizard, you set the size of the building that contains your office space. You also set the thickness of the walls.

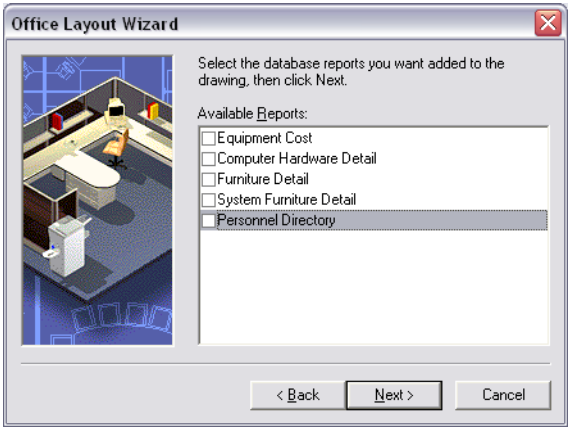


3 Do the following:

- In the row of office shell buttons, click the square shape (the first shape).
- In the A box, enter **25**' for the length.
- In the B box, enter **15**' for the width.
- In the Wall Thickness box, enter **0**' **6**".

4 Click Next.

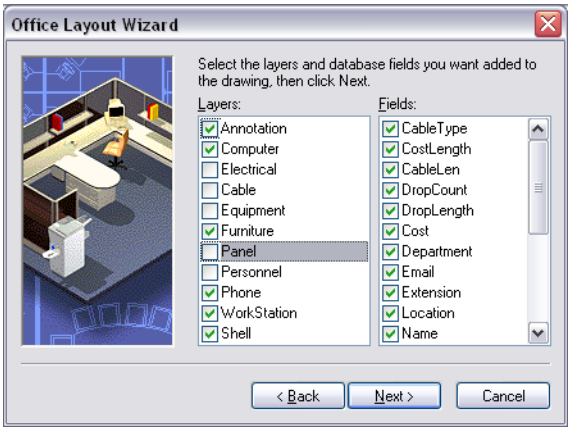
The next page of the wizard lists database reports that you can add to your drawing.



5 Clear all of the check boxes on this page. (You don't need database reports in this tutorial.) Click Next.

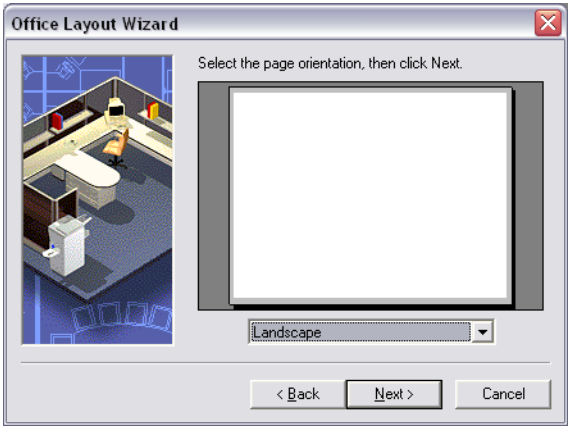
On the next page of the wizard, select the layers for the drawing and the fields in the database.

Note Because you did not select any database reports on the previous wizard page, AutoSketch will ignore the Fields settings. So, you can leave the database fields as they are preset on this page.



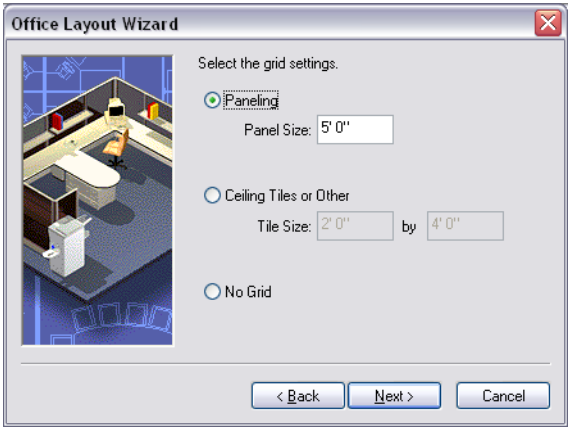
6 In the Layers list, clear the check boxes next to Electrical, Cable, Equipment, Panel, and Personnel. Then, click Next.

Now, choose the page orientation that best displays the office layout.



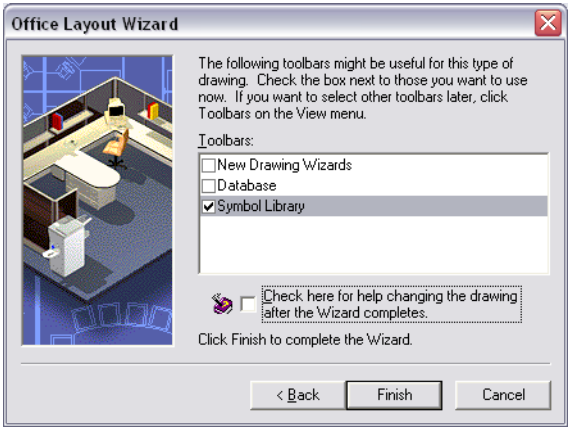
7 Select Landscape from the list, and then click Next.

On the next page of the wizard, you can select a grid setting. Because you will set a specific grid setting in the next exercise, you can accept the pre-selected setting (Paneling) on this page.



8 Click Next.

Now, choose a toolbar to display in the drawing window.

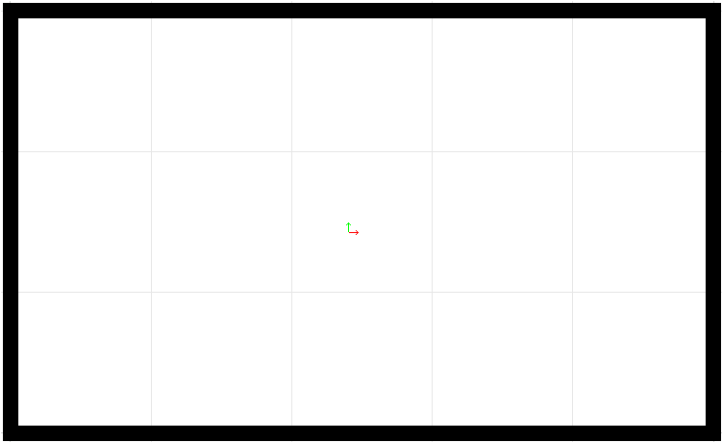


9 In the Toolbars box, click the check box next to Symbol Library. You will use that toolbar later.

10 Below the Toolbars box, clear the check box next to the Help icon. Then, click Finish.

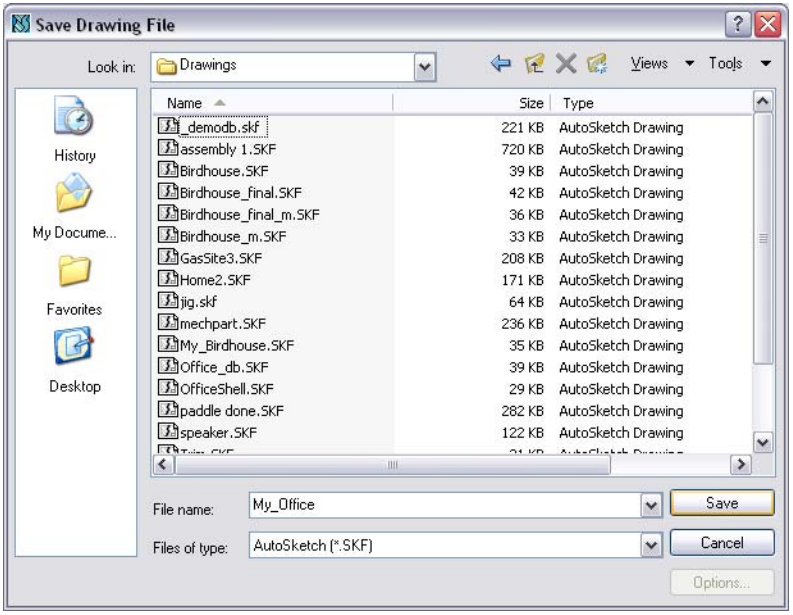
You have just completed the Office Layout wizard.

In the AutoSketch window, the following drawing is displayed.



11 On the File menu, click Save.

12 In the Save Drawing File dialog box, navigate to the following location.
C:\Program Files\Autodesk\AutoSketch9\Drawings



13 In the File Name box, enter **My_Office**, and then click Save.
Next, set units, a reference grid, and the scale.

Set Units, Reference Grid, and Scale

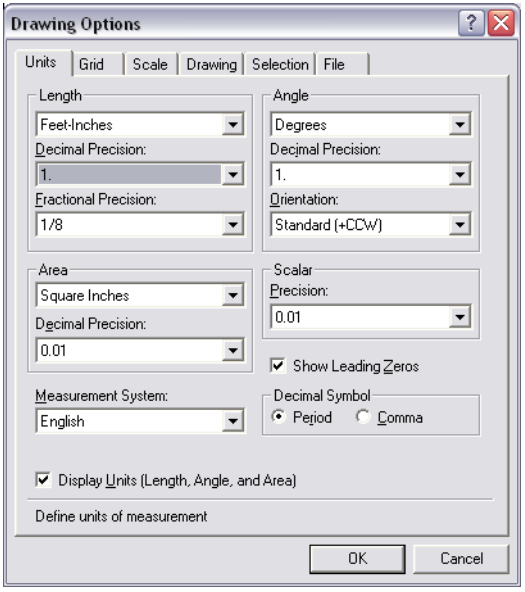
In this exercise, you learn to

- ❒ Right-click to access the Drawing Options dialog box.
- ❒ Set units, reference grid, and scale in the Drawing Options dialog box.

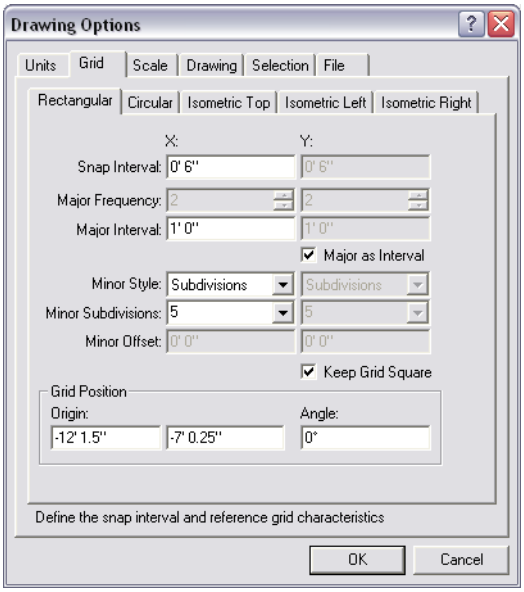
The office layout drawing you are creating requires some adjustment of the preset units, reference grid, and scale settings. You can adjust all of these settings in the Drawing Options dialog box.

To set units, reference grid, and scale

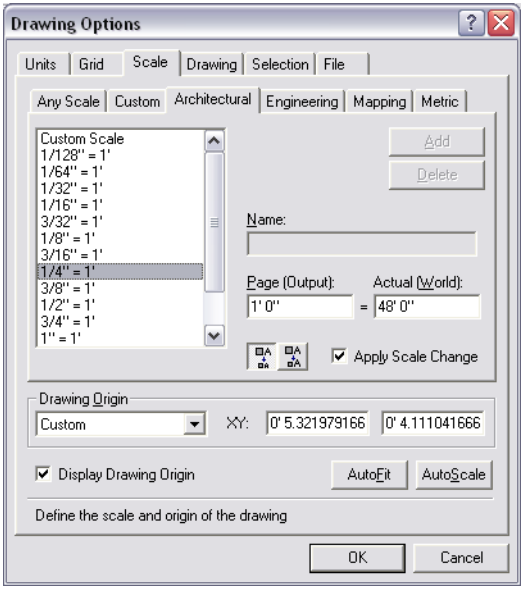
1. In a blank area of the drawing, right-click, and then click Drawing Options.
2. In the Drawing Options dialog box, Units tab, under Length, click the arrow to the right of the Decimal Precision box, and select 1. from the list.



3. On the Grid tab, select the Rectangular subtab (as shown in the following illustration), and then change these settings:
   - In the Snap Interval box, enter **6**".
   - In the Major Interval box, enter **1**'.

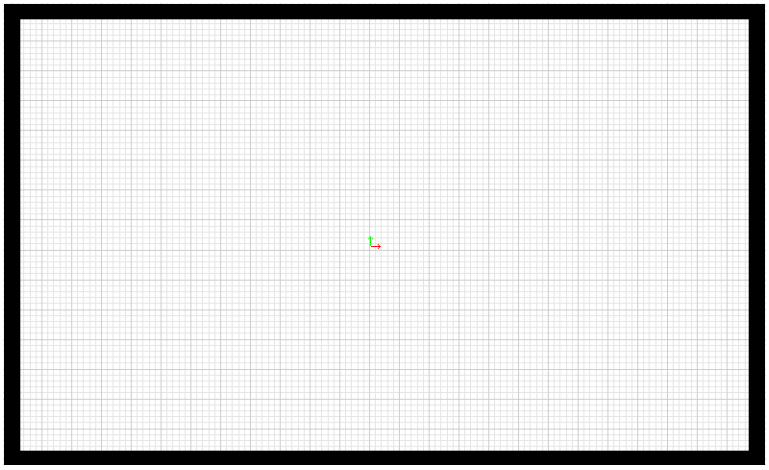


4 On the Scale tab, select the Architectural subtab (as shown in the following illustration). In the list of scale settings, select 1/4" = 1'.



5 Click OK to close the Drawing Options dialog box and save your changes.

The drawing should now match the grid settings in the following illustration.



Now that you have set the units, reference grid, and scale of the office layout, you can create the partition walls of the office.

Create Office Partition Walls

In this exercise, you learn to

- ❒ Add a partition wall to the office space.
- ❒ Zoom to display all entities in the drawing using the Extent command.
- ❒ Choose a corner polyline using the Wizard Tools toolbar.
- ❒ Set the width and lengths of the walls using the edit bar.

To create office partition walls

1. On All-In-One toolbar, click and hold the Zoom button.

2. On the Zoom toolset, drag the pointer to select View Extent.

   The drawing window now includes the entire office shell.

3. On the Wizard Tools toolbar, click Corner Polyline.

   The edit bar displays tools related to editing a corner polyline. The pointer in the drawing window changes to a hollow circle.

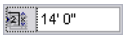
4 On the edit bar, in the Width box, enter **0' 6"**.

This sets the wall width to match the one you selected for the building in the Office Layout wizard.

Now, set the length of the first wall.

5 On the edit bar, in the Corner Length 1 box, enter **10' 8"**.

Then, set the length of the second wall.

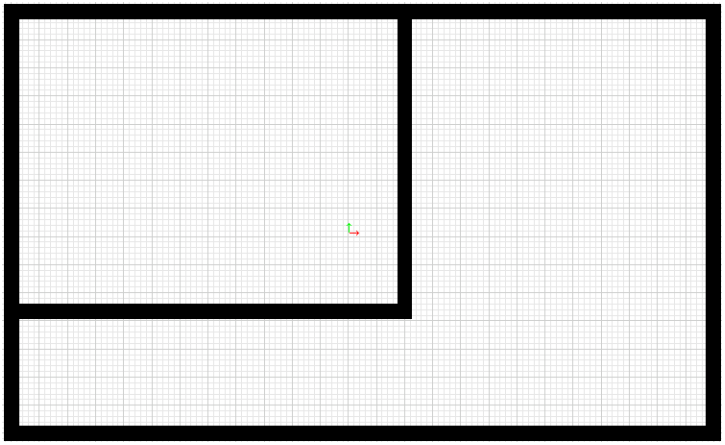


6 On the edit bar, in the Corner Length 2 box, enter **14'0"**.

7 In the drawing, click the wall on the left side, and then click the wall at the top to place the wall.

8 Right-click to end the command.

The interior walls you defined using the edit bar are now displayed in the drawing.



9 Save your work.

Now, it's time to create the doors and windows in the office layout.

Add Doors and Windows

In this exercise, you learn to

- ❒ Choose symbols by using the Wizard Tools toolbar and the Content Librarian.
- ❒ Add doors and windows to precise locations in your drawing.

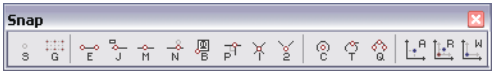
Up to this point, you have created your own lines, arcs, polygons, and other entities to represent elements in your drawings.

In this exercise, you learn how to use just a few of the thousands of predrawn entities, called *symbols*, that are available to you in AutoSketch. You can see how powerful this feature is by using symbols in your office layout.

To add doors and windows

1 On the Snap toolbar, click the Snap button to turn off Snap settings.

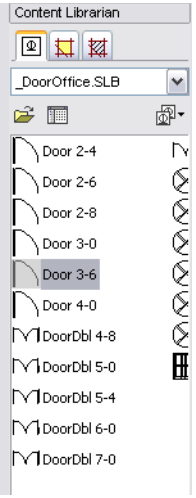
Note When Snap settings are turned off, all of the buttons in the toolbar are deselected, as shown in the following illustration.



You won't need to use precision snap settings in this exercise.

2 On the Wizard Tools toolbar, click the Doors button.

On the right side of the AutoSketch window, the Content Librarian displays thumbnail images of different door sizes and types.



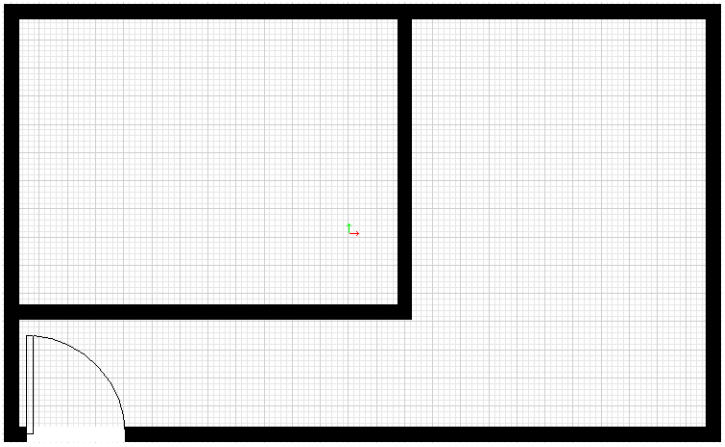
3 In the Content Librarian, locate Door 3-6 (a door that is 3 ft, 6 inches wide).

4 Click Door 3-6 and drag it into the drawing. When the symbol is close to the bottom-left corner of the drawing, release the mouse button. Then, move the pointer up slightly, and click to place the door so that it opens into the office space.

Tip You can always "undo" an action, like the placement of the door, by pressing CTRL+Z on the keyboard.

5 Right-click to end the command.

Your drawing should match the following illustration.

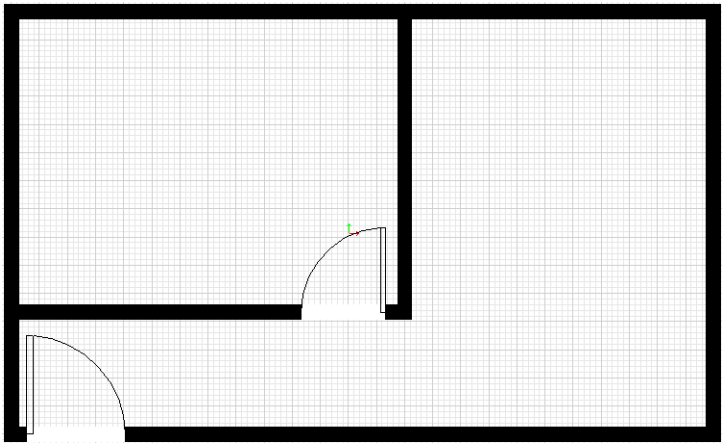


Next, add a door to the interior office.

6 In the Content Librarian, click and hold Door 3-0, and drag it into the drawing. When the vertical part of the symbol is close to the bottom-right corner of the interior office, release the mouse button. Then, move the pointer up and to the left slightly, and click to place the door so that it opens into the interior office space.

7 Right-click to end the command.

Your drawing should match the following illustration.



8 Save your work.

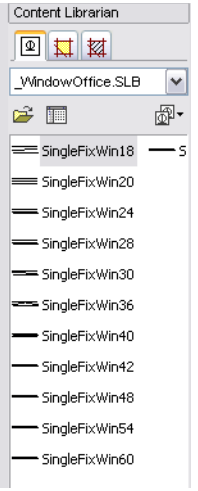
Now, add some windows to the office space.

To add windows

1. On the Wizard Tools toolbar, click and hold the Doors button.

2. On the toolset, drag the pointer to select Windows.

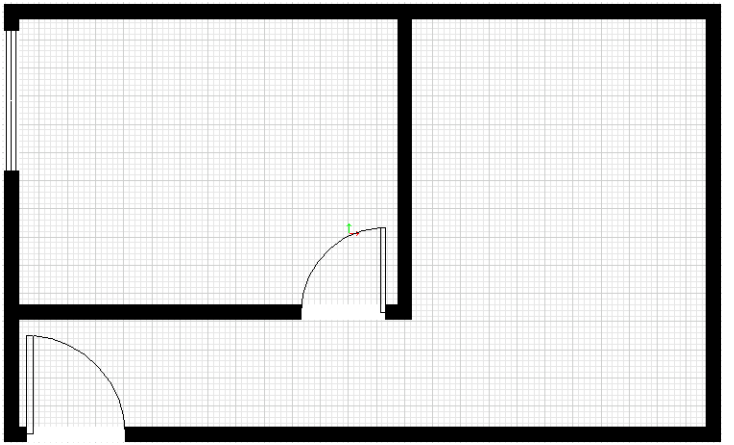
   On the right side of the AutoSketch window, the Content Librarian displays thumbnail images of different office-window sizes and types.



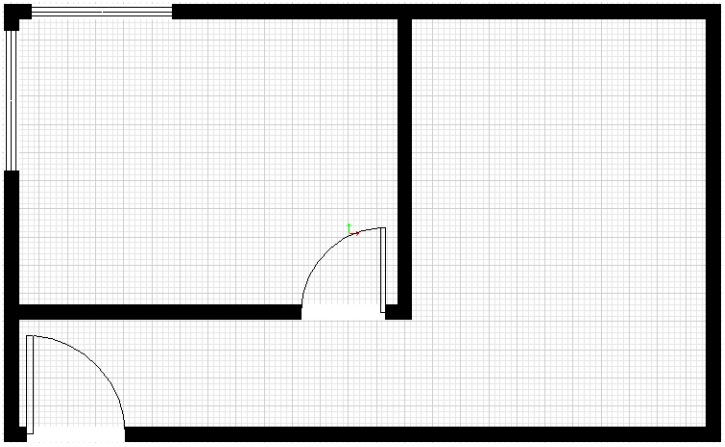
3. In the Content Librarian, locate SingleFixWin60.

   You add this window type to two locations in your drawing.

4. Click and hold SingleFixWin60, and drag it on the side wall at the upper left, as shown in the following illustration.



5 With the symbol still active, click the top wall at the upper left to place the second window there, as shown in the following illustration. Then, right-click to end the command.

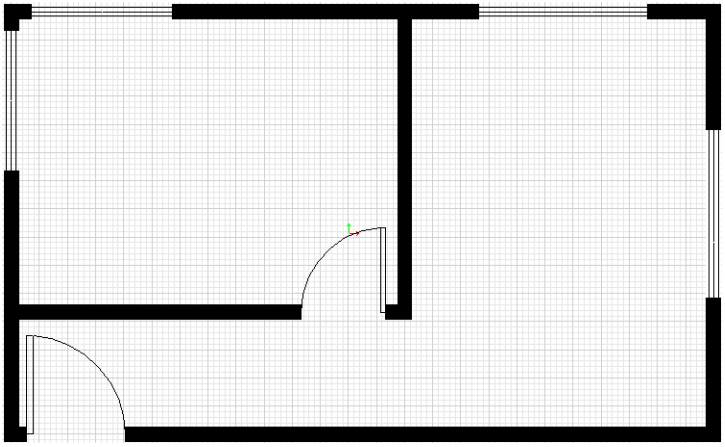


Next, add a slightly larger window to two locations outside the internal office.

6 In the Content Librarian, click and hold SingleFixWin72, and drag it to the top wall, close to the right side. Click again to place the symbol.

7 Click the vertical wall close to the top. Then, right-click to end the command.

Your drawing should look similar to the following illustration.



8 Save your work.

You are ready to add the furniture to the office layout.

Add Furniture

In this exercise, you learn to

- ❒ Choose symbols by using the Wizard Tools toolbar and the Content Librarian.
- ❒ Add a workstation symbol to the interior office.
- ❒ Move and rotate a symbol.
- ❒ Place symbols in precise locations using the Absolute Coordinates dial.

Once the interior walls, doors, and windows are in the drawing, you can add a workstation and other furniture to the office.

1 On the Wizard Tools toolbar, click the Private Offices button.

On the right side of the AutoSketch window, the Content Librarian displays thumbnail images of different private offices.

2 In the Content Librarian, locate RPrivOffProSta.

The symbol is an office workstation.

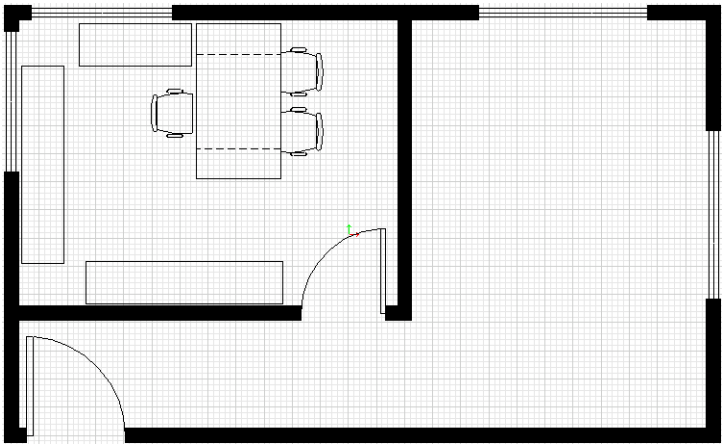
3 Click RPrivOffProSta and drag it into the interior office section of the drawing. Do not release the mouse button yet.

4 On the keyboard, press the F5 key until the office workstation is rotated counter-clockwise and the desk and chairs are on the right side in the interior office. (SHIFT+F5 rotates the workstation clockwise.)

Tip You can also use the numeric keypad on the right side of the keyboard to rotate entities. To rotate entities clockwise, press the - (minus) sign. To rotate entities counter-clockwise, press the + (plus) sign.

5 Click to place the workstation, and then right-click to end the command.

Your drawing should look similar to the following illustration.



You have rotated and placed the workstation in the interior office space.

6 On the File menu, click Save.

You are ready to create a table and chairs for the outer office.

Create a Round Table and Chairs

In this exercise, you learn to

- ❒ Create a round table using the All-In-One toolbar and the edit bar.
- ❒ Set a precise radius using the edit bar.
- ❒ Use the Absolute Coordinates dial to place items in precise locations.
- ❒ Rubber-stamp an existing chair and place it in another location.
- ❒ Create a circular array of four additional chairs that are placed around the table.

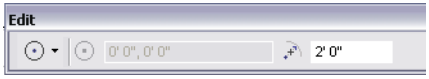
To create a round table and chairs

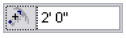
1 On the Snap toolbar, click Snap to turn on snap settings (you turned them off earlier, when you placed the doors and windows).

2 On the All-In-One toolbar, click and hold the Circle button.

3 On the toolset, drag the pointer to select Center, Radius Circle.

The edit bar displays tools related to editing a center, radius circle. The pointer becomes a hollow circle with four points.

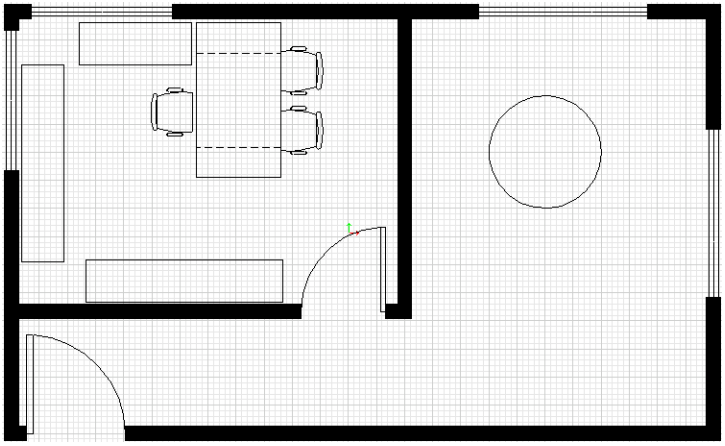




4 On the edit bar, set the radius to 2'0".

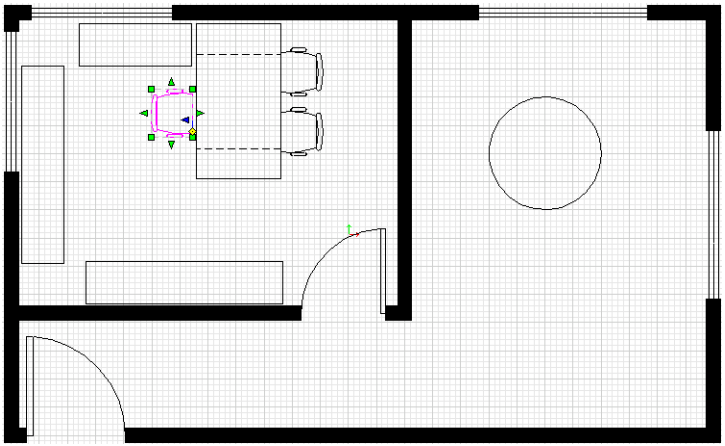
5 On the right side of the drawing, click when the Absolute Coordinates dial shows 19'0", 10'0". Then, right-click to end the command.

Your drawing should match the following illustration.



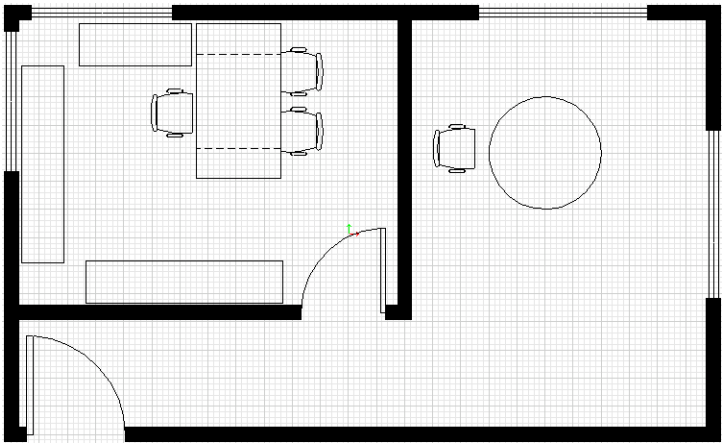
Now, place five chairs around the round table. Instead of creating each chair individually, create one chair by rubber-stamping an existing chair. Then, create a *circular array* (a selection set arranged in a circular pattern) of four chairs, arranging them around the table.

6 In the interior office space on the left side of the drawing, click the chair that faces the other two chairs.

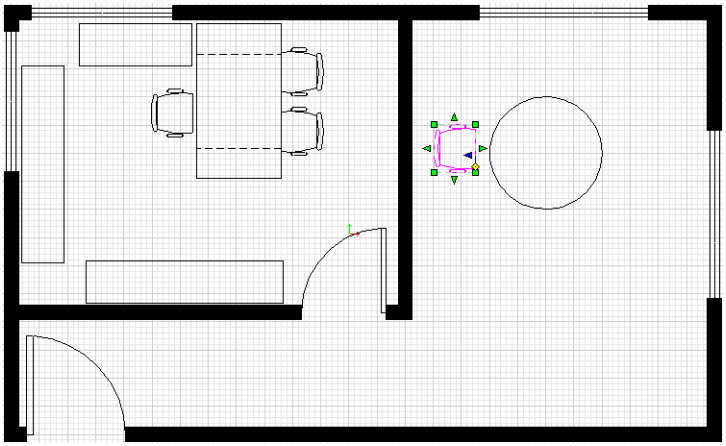


7 On the All-In-One toolbar, click Rubber Stamp. Then, drag the rubber-stamped chair, and when the Absolute Coordinates dial reads 16.6", 9.6", click to place the chair next to the round table. Then, right-click to end the command.

Your drawing should match the following illustration.



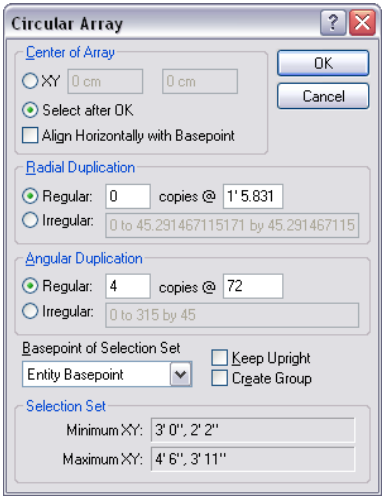
8 Click the chair you just created to select it.



9 On the All-In-One toolbar, click and hold the Rubber Stamp button.

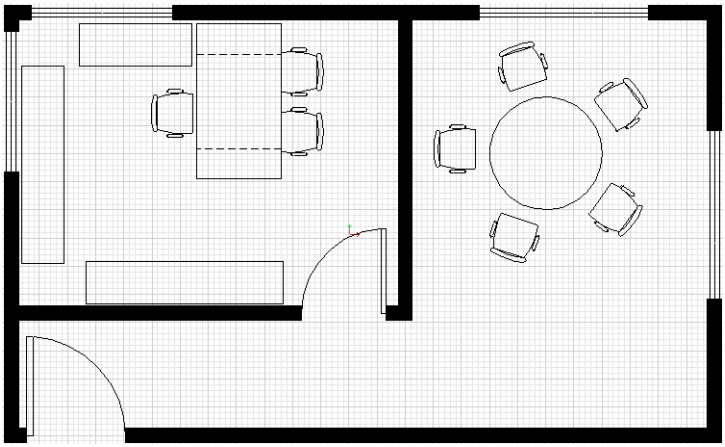
10 On the toolset, drag the pointer to select Circular Array.

11 In the Circular Array dialog box, in the Angular Duplication area, select the Regular option, and enter **4** in the first box, and **72** in the second. Then, click OK.



12 In the drawing window, move the pointer over the edge of the table until the Centerpoint snap is displayed. Then, click to add the arrayed chairs.

Your drawing should match the following illustration.



Great work! You have completed the third tutorial. You should now be able to create entities, place symbols, set dimensions, add layers, use wizards, and create arrays.

You can finish this drawing by setting dimensions, adding a title to the drawing, and placing the entities on appropriate layers. Use what you learned in the second tutorial, "Create a Birdhouse Drawing," to complete this drawing.

Remember that you can always get detailed information from the Help system by clicking AutoSketch Help on the Help menu.

Tutorial 4 — Advanced Exercises

In this tutorial, you learn how to create 3D effects, use the Web tools called eTransmit, hyperlinks, and Communication Center, and generate a database report.

More information about each of the concepts in this tutorial is available in the AutoSketch® Help system.

In this tutorial

- Create 3D Effects
- Use Web Tools
- Generate a Database Report

Create 3D Effects

In this exercise, you learn to

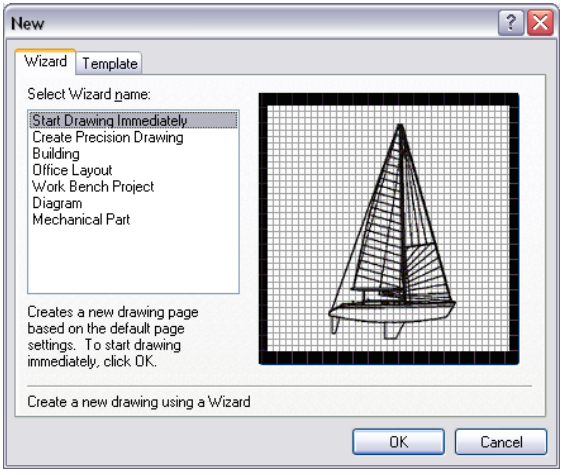
- ❒ Create a 3D parallel extrusion.
- ❒ Use the Standard toolbar.
- ❒ Use the 3D Effects toolbar.

While two-dimensional drawings are the basic building blocks of most projects, you might need a three-dimensional view of a project. In AutoSketch, you can simulate three-dimensional drawings through *parallel* and *perspective* extrusion.

- 3D parallel extrusion creates a copy of the selection set that you place in the drawing. AutoSketch connects corresponding edges with lines or polygons. You learn how to create a 3D parallel extrusion in this exercise.
- 3D perspective extrusion creates a scaled copy of the selection set. You place the copy anywhere in the drawing, and AutoSketch connects the corresponding edges using lines or polygons.

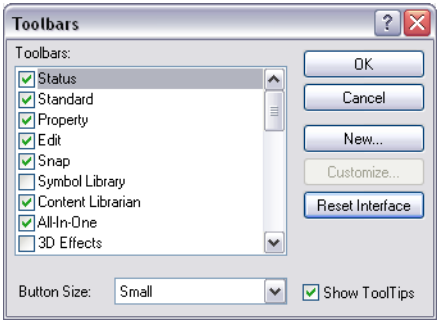
To apply parallel extrusion to an entity

1 On the File menu, click New. In the New dialog box, click Start a Drawing Immediately, and then click OK.



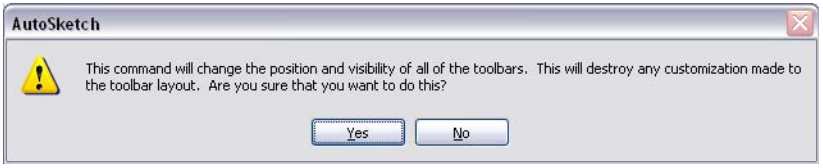
Now, you reset the interface to make sure your settings match the tutorial's instructions.

2 On the View menu, click Toolbars.



3 In the Toolbars dialog box, in the lower-right corner, click Reset Interface.

4 In the warning message that is displayed, click Yes.



Your drawing window is now opened and set up for this tutorial.

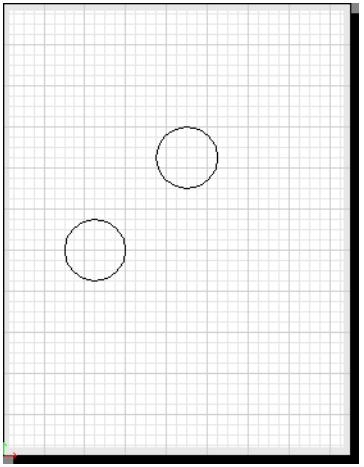
5 On the Snap toolbar, click the Centerpoint Snap button to turn it on.

6 On the All-In-One toolbar, click the Circle button.

7 On the toolset, drag the pointer to select Center, Side Circle.

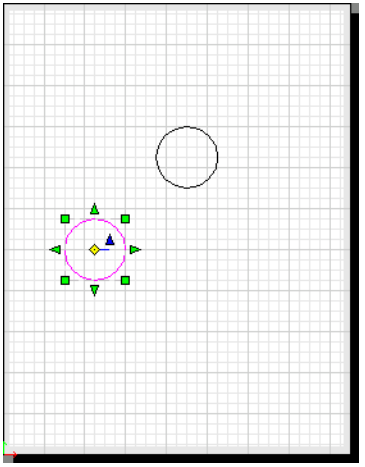
8 In the drawing, create two circles of roughly the same size.

Your drawing should look similar to the following illustration.



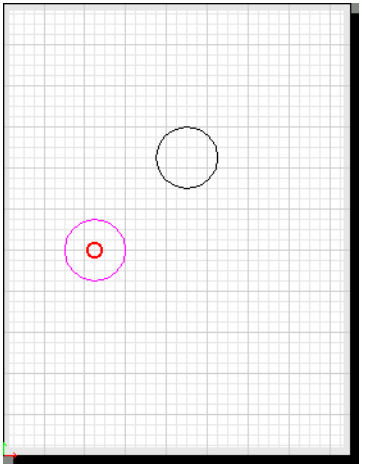
9 On the Standard toolbar, click the 3D Effects button.
The 3D Effects toolbar is displayed.

10 Click the circle that is closer to the bottom of the drawing.

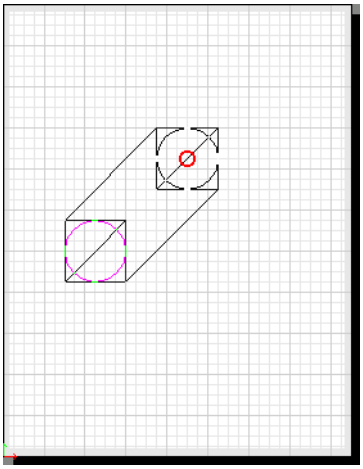


11 On the 3D Effects toolbar, click the 3D Parallel Extrusion button.

12 In the drawing, click the bottom circle again.



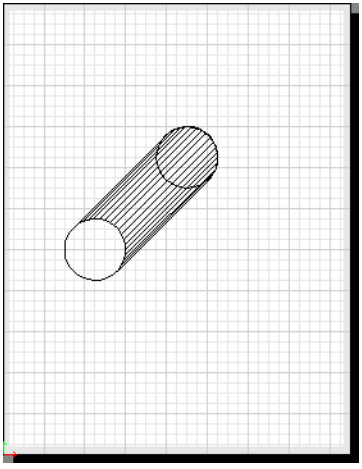
This is the first of two points that define the distance and direction that the selection set will be extruded. A rubber-band extrusion appears and moves the same distance and direction as the pointer.



13 Click the edge of the second circle. The selection set is extruded.

14 Right-click to end the command.

Your drawing should look similar to the following illustration.



15 On the File menu, click Close.

16 In the Save Changes to Drawing dialog box, click No.

Use Web Tools

AutoSketch Web tools include Web editing tools (such as eTransmit and hyperlinks creation) and the Communication Center.

Use eTransmit

In this exercise, you learn to

- ❒ Create a transmittal set of an AutoSketch drawing and related files.
- ❒ Email the transmittal set (an Internet connection is required).
- ❒ Check the receipt of the transmittal set and save it (an Internet connection is required).

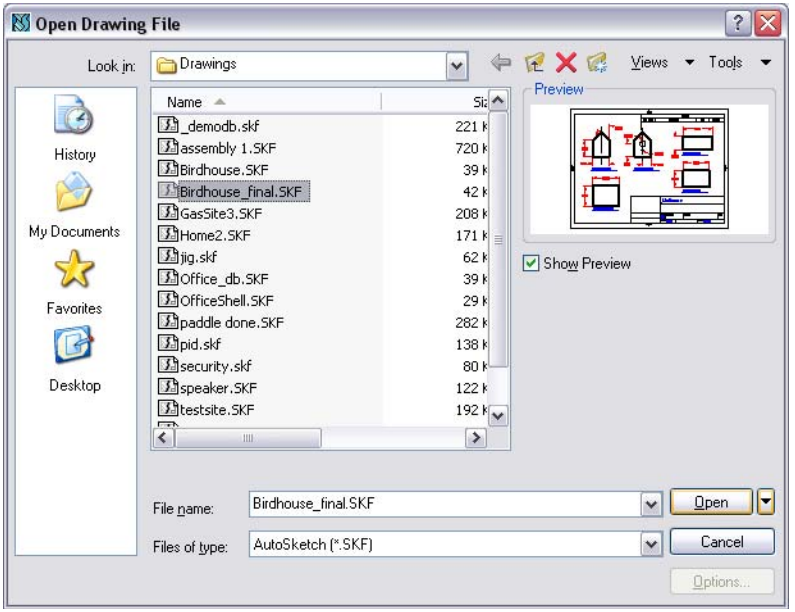
With eTransmit, you can create a transmittal set of an AutoSketch drawing that automatically includes all related files. You then publish the transmittal set as an email attachment.

When you use eTransmit, a report file is automatically generated that includes instructions detailing what files are included in the transmittal set and what must be done with them so that they are usable by the original drawing. You can also do the following:

- Add notes to the report and specify password protection for the transmittal set.
- Specify path options for the transmittal set and create a self-extracting executable or zip file that packages all the files.

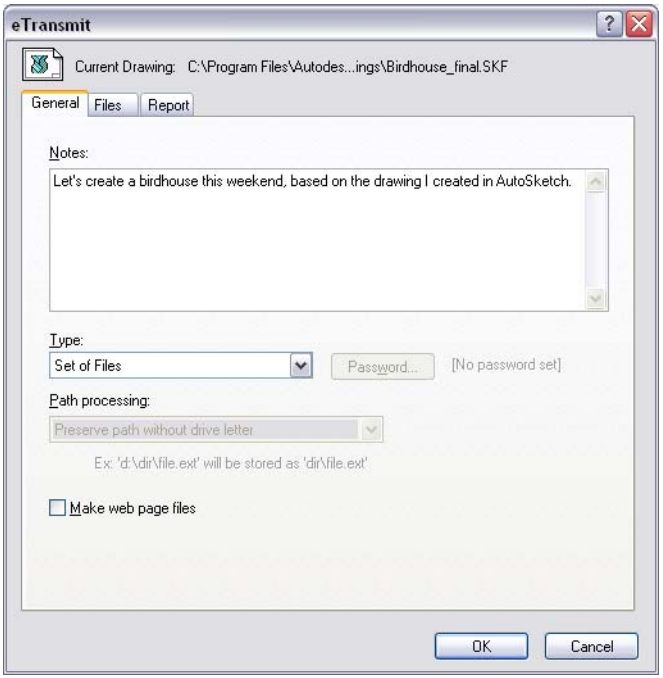
To create a transmittal set

1 On the File menu, click Open.



2 In the Open Drawing File dialog box, in the *Drawings* folder, click *Birdhouse_final.skf*. Then, click Open. (This drawing is the completed drawing that you worked on in Tutorial 1, "Create a Birdhouse Drawing.")

3 On the File menu, click eTransmit.



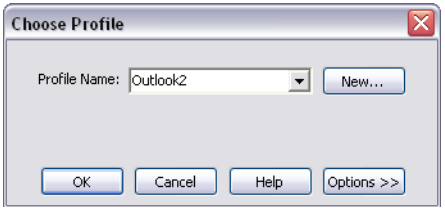
4 In the eTransmit dialog box, in the Notes section, add any notes you want.

5 In the Type list, select Set of Files.

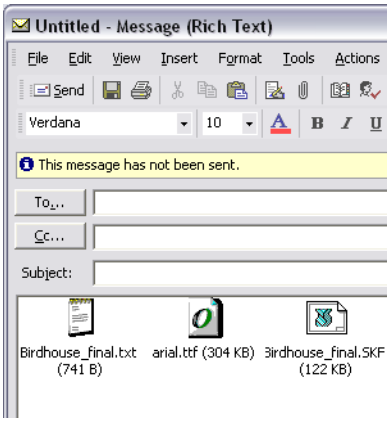
6 Clear the check box next to Make Web Page Files. You don't make a Web page in this exercise.

7 Click OK.

8 If the Choose Profile dialog box is displayed, click OK.



Your default email program opens with an email that contains the files, as shown in the following illustration.



9 In your email program, enter your own email address, and send the email.

10 Check your email inbox, and save the transmitted files.

11 In AutoSketch, on the File menu, click Close. Do not save your work.

Create a Hyperlink

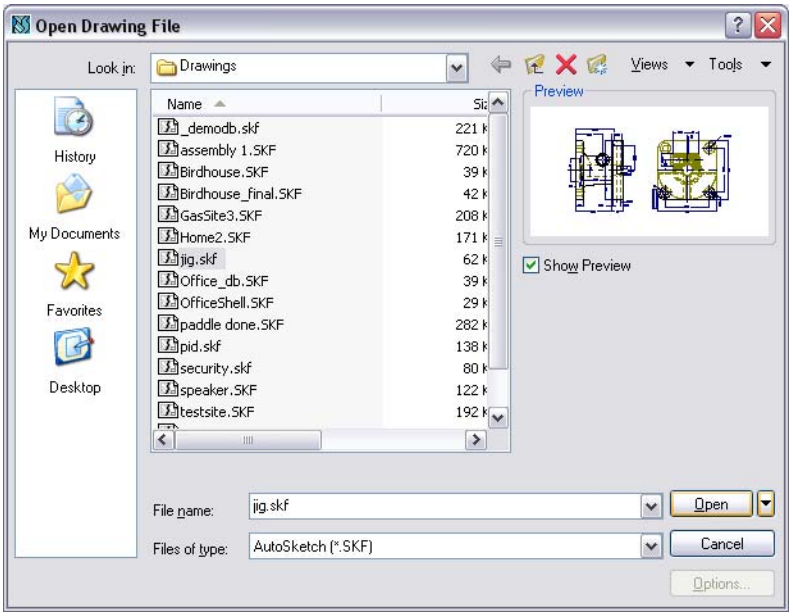
In this exercise, you learn to

- ❒ Assign a hyperlink to a Web site.
- ❒ Open the Web site (an Internet connection is required).

You can assign hyperlinks to entities in your drawings that point to Web pages, or to other files stored on local servers or on the Internet.

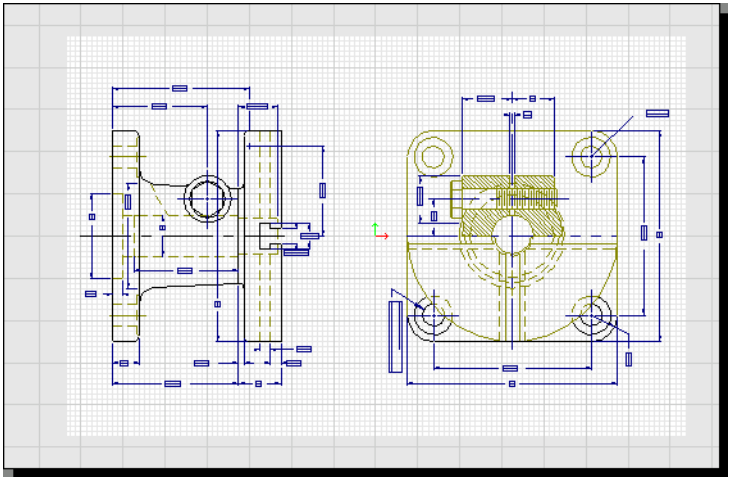
To assign a hyperlink to a Web site

1 On the File menu, click Open.



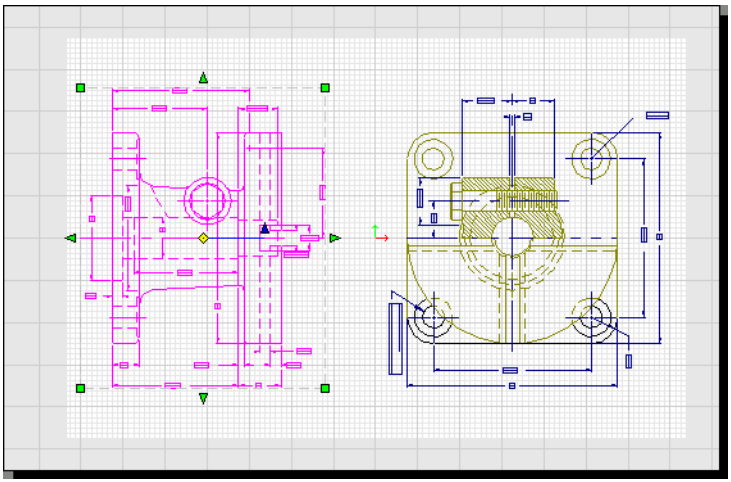
2 In the Open Drawing File dialog box, in the *Drawings* folder, click *Jig.skf*. Then, click Open.

The drawing you opened should match the following illustration.



3 Click and drag your pointer from the top left portion of the illustration on the left to the lower-right corner of that illustration. When all of the entities in the illustration on the left are selected, release the mouse.

The drawing should match the following illustration.



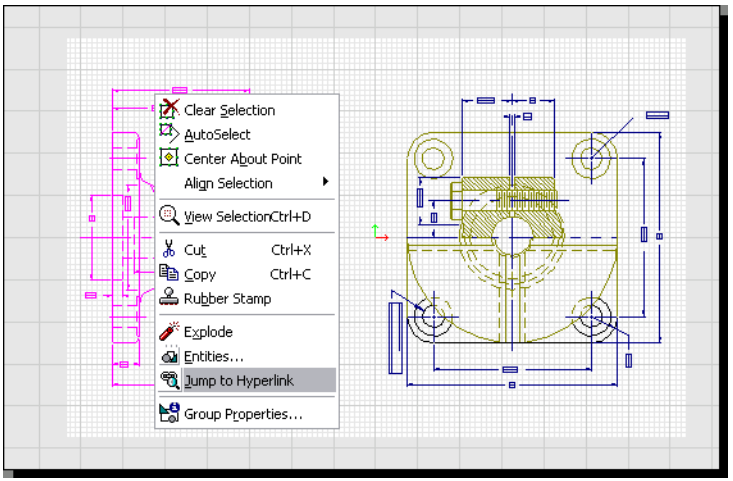
4 With the entity still selected, right-click, and then click Group. Do not right-click again, because you want the entity to remain selected.

The individual entities are now grouped into a single entity. Now, assign a hyperlink to this grouped entity.

5 On the Database menu, click Edit.

6 In the Edit Hyperlink dialog box, in the Hyperlink (File or URL) text box, enter **www.autodesk.com**, and then click OK.

The entity you grouped is now hyperlinked to the Autodesk Web site.



7 Anywhere in the grouped entity, right-click, and then click Jump to Hyperlink.

If you have an Internet connection, the Web site now opens.

Use the Communication Center

In this exercise, you learn to

- ❒ Open the Communication Center.
- ❒ Set up the Communication Center.

The Communication Center provides the following kinds of announcements:

- **General Product Information.** Stay informed about Autodesk company news and product announcements; give your feedback directly to Autodesk.
- **Product Support Information.** Get breaking news from the Product Support team at Autodesk.
- **Articles and Tips.** Be notified when new articles and tips are available on Autodesk Web pages.

To set up the Communication Center

1 If AutoSketch is not already open, start it now.

2 In the AutoSketch window, click the Communication Center icon, located on the right side of the status bar.

3 In the Communication Center window, specify your country or region.

4 Specify the frequency you prefer for updates.

5 Specify the information channels you want displayed.

Now that you have set up the Communication Center, you will start to receive updates, based on the frequency you set. You can always change the frequency with which you receive updates by clicking the Communication Center icon, and then clicking the Settings button.

Generate a Database Report

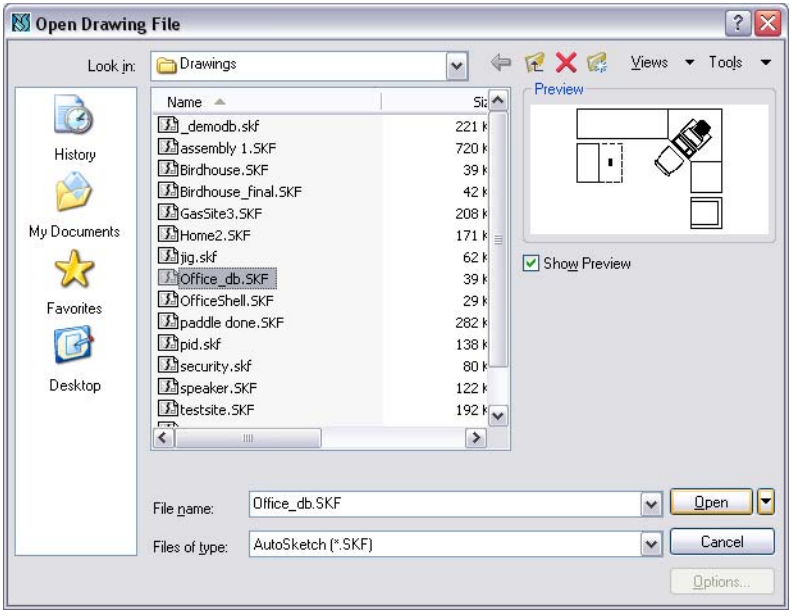
In this exercise, you learn to

- ❒ View a symbol's properties.
- ❒ Select a database report.
- ❒ Run a report.
- ❒ Print a report.

The office plan you use for this exercise is a simple office plan, much like the office layout you created in an earlier tutorial. This drawing contains several symbols, each of which represents individual office items that can be ordered from a manufacturer. Using the database feature in AutoSketch, you can create a database report to run when the time comes to order the items.

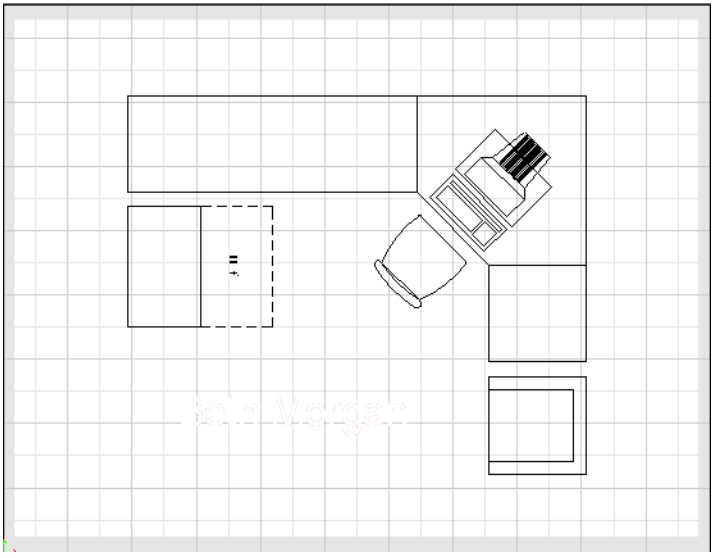
To create a database report

1 On the File menu, click Open.

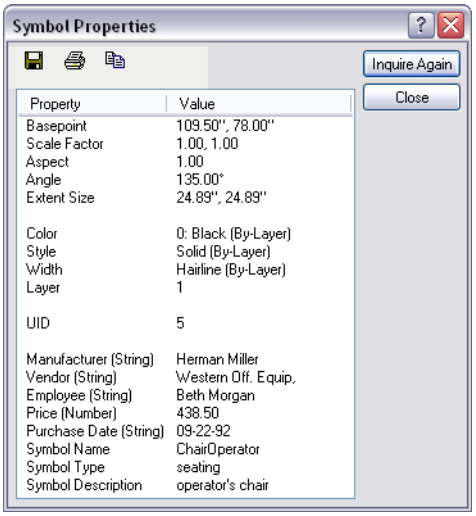


2 In the Open Drawing Files dialog box, in the *Drawings* folder, click *Office_db.skf*. Then, click Open.

The drawing you opened should match the following illustration.



3 In the drawing window, right-click the chair, and then click Symbol Properties to display the Symbol Properties dialog box.

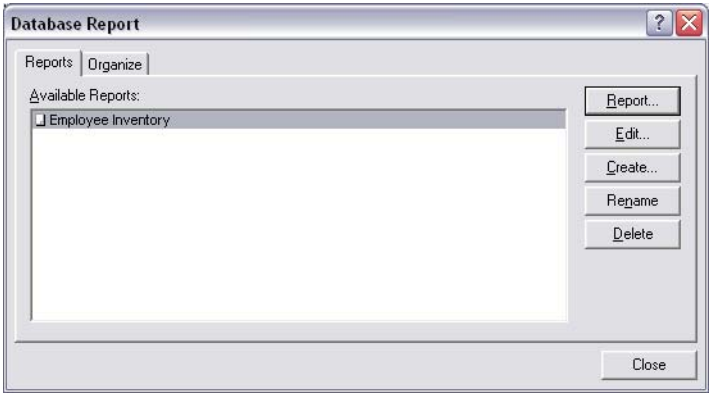


The chair symbol contains a lot of information. Viewing the Symbol Libraries dialog box, you can find out this symbol's properties and values, such as its basepoint, scale factor, color, width, layer, name, and manufacturer. This information can be extracted to a database report.

4 In the Symbol Properties dialog box, click Close.

5 In the drawing, press CTRL-A to select all entities.

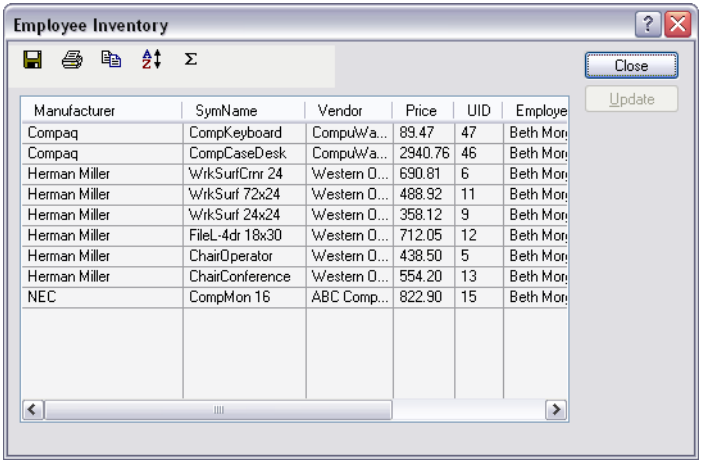
6 On the Database menu, click Report.



In the Database Report dialog box, you can see that one report, "Employee Inventory," is selected.

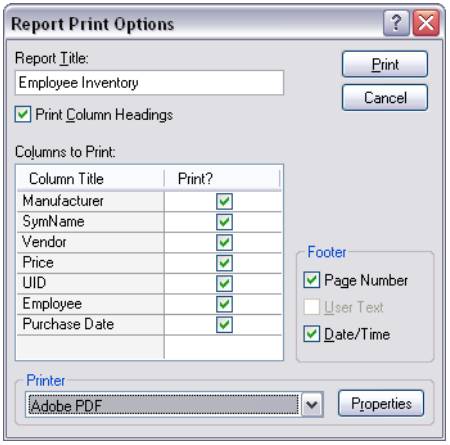
7 Click Report.

The Employee Inventory report is generated, as shown in the following illustration.



8 In the Employee Inventory window, on the toolbar, click the Print button.

9 In the Report Print Options dialog box, click Print to print the Employee Inventory report.

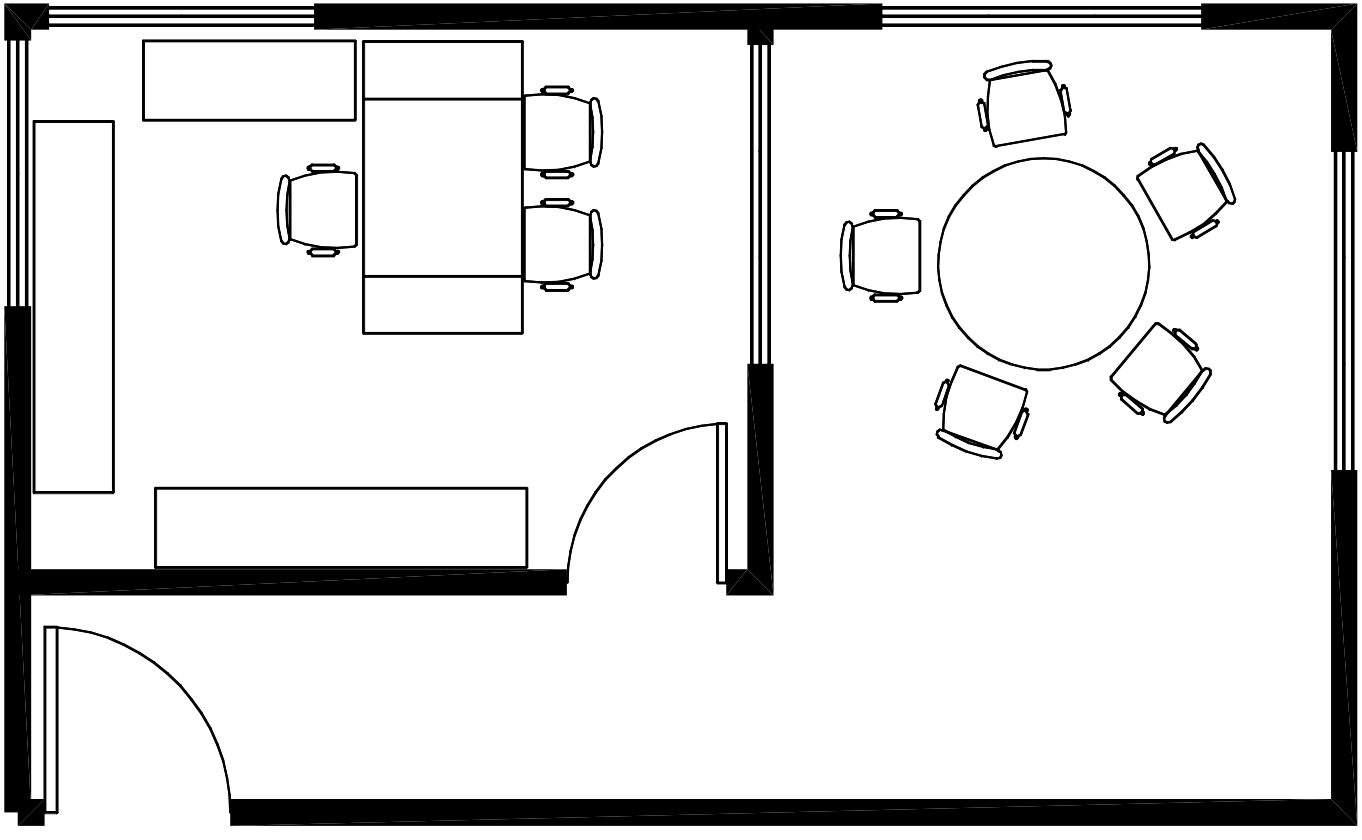


Nice work! You have completed all of the tutorials offered in *Getting Started*. Now, you can become more proficient with CAD, explore AutoSketch, and use the program for your drawing needs.

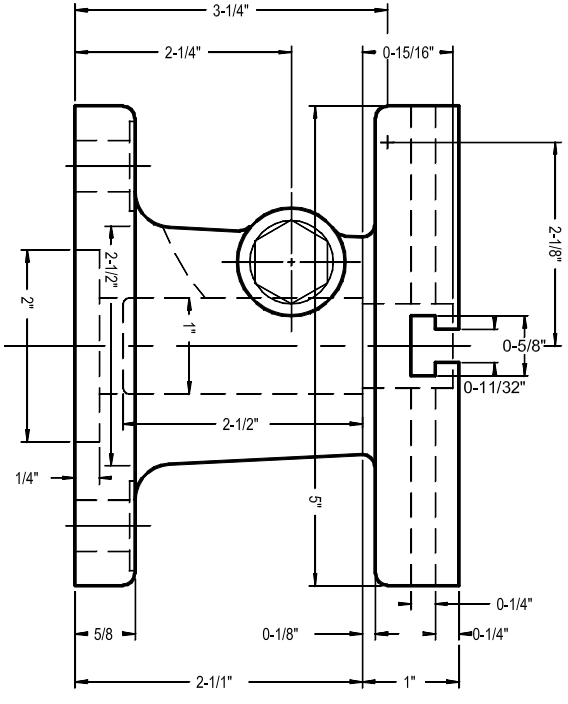
To find out how to perform tasks not highlighted in these tutorials, or to learn more conceptual information, see the AutoSketch Help system.

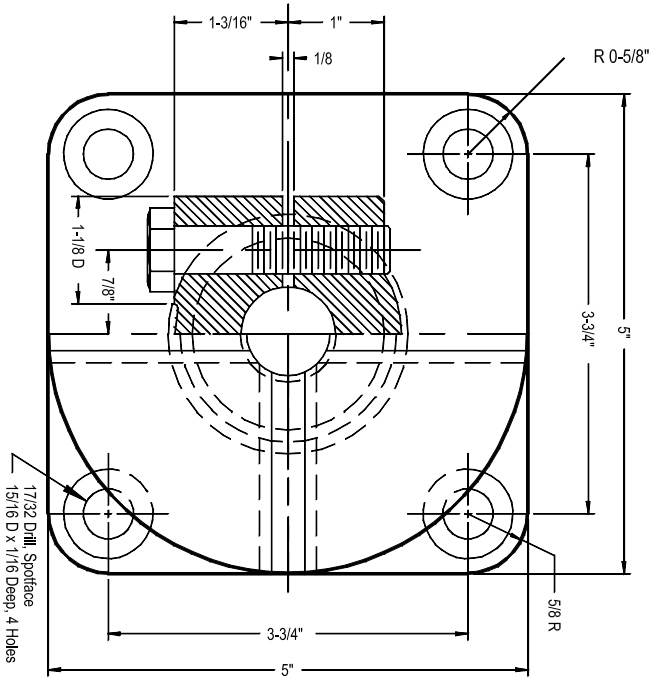
Appendix — Drawings Created with AutoSketch

This appendix contains several drawings that were created with AutoSketch®. Study these drawings to get ideas for your own drawings, or just to see the power of the product.

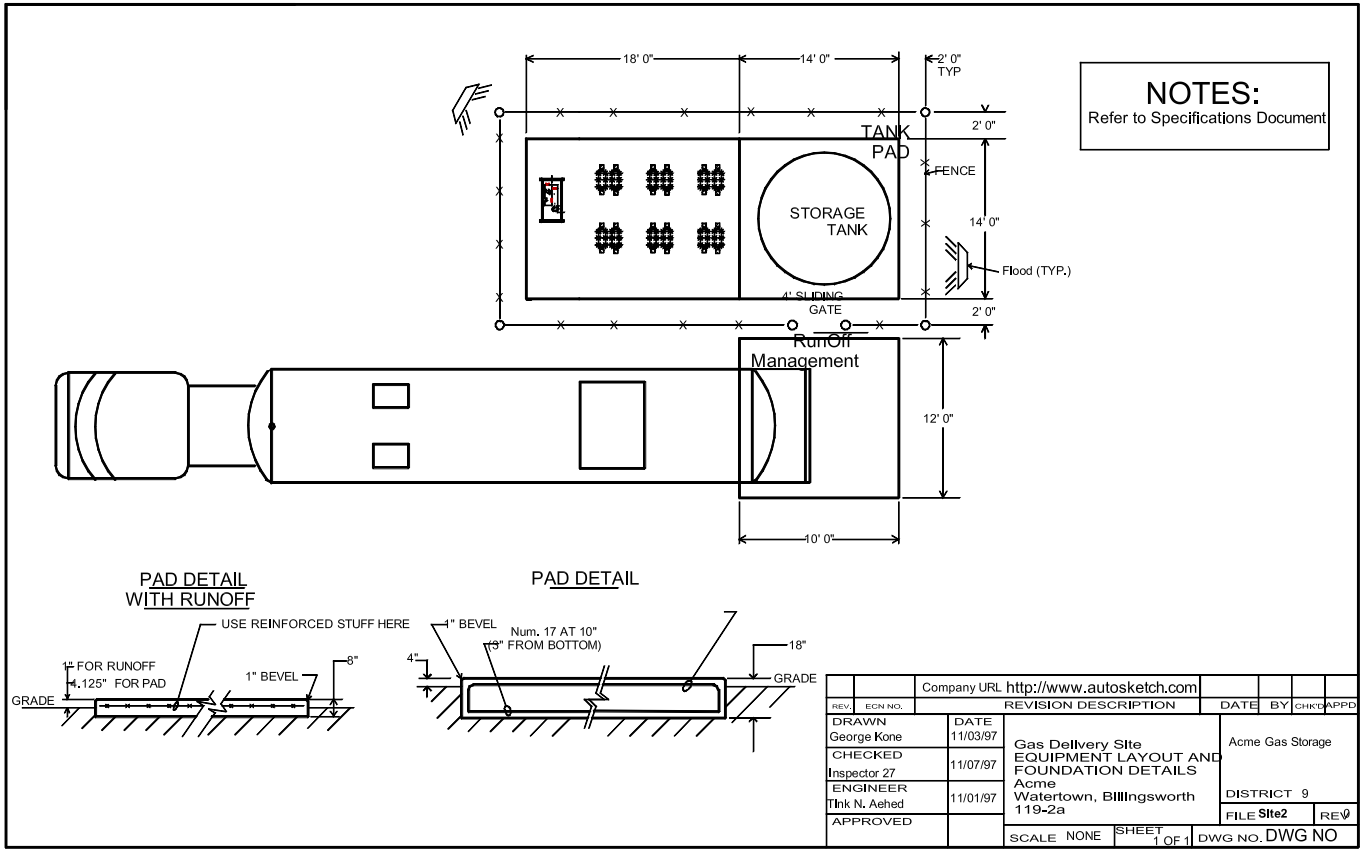


Office – created using the Office Layout wizard

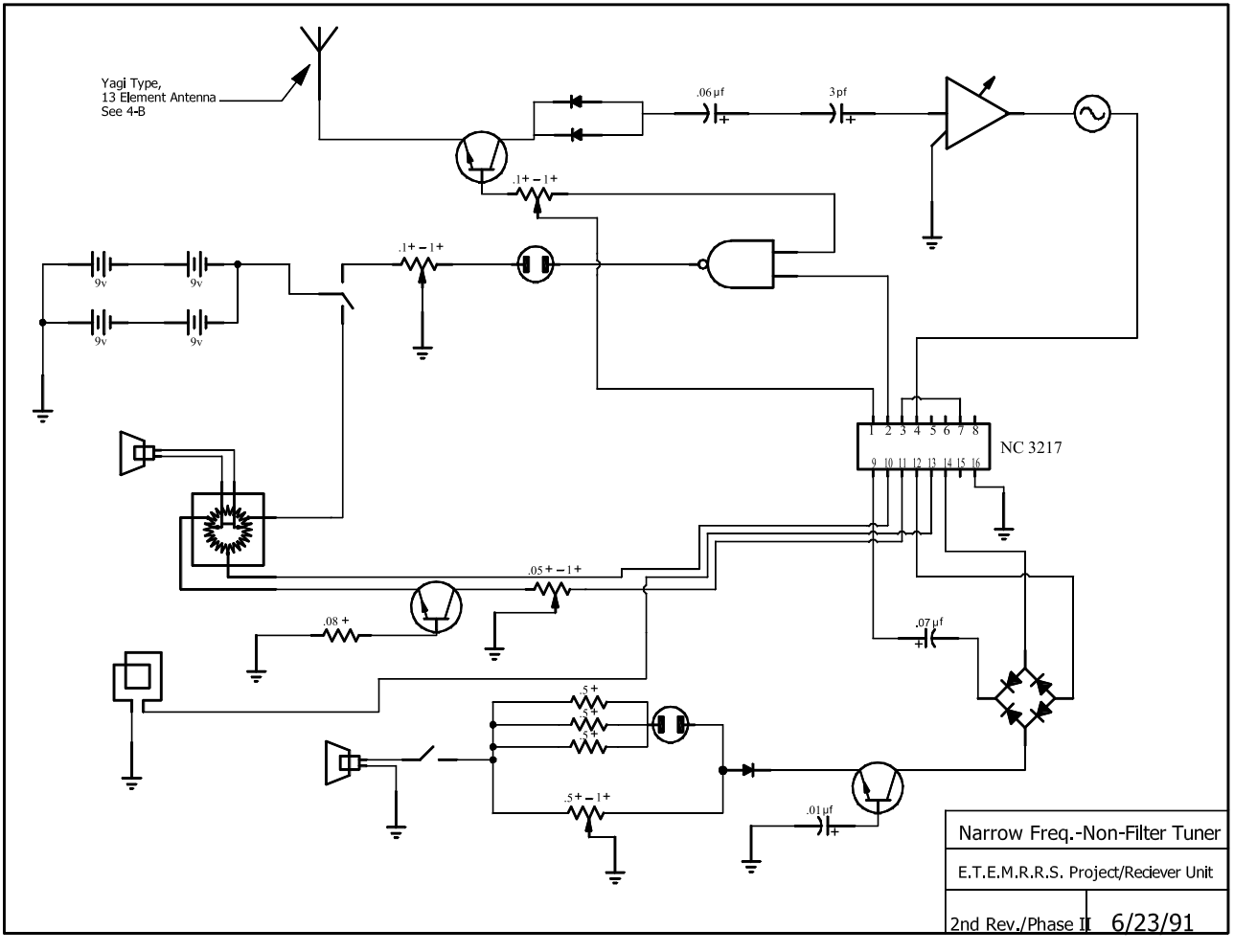




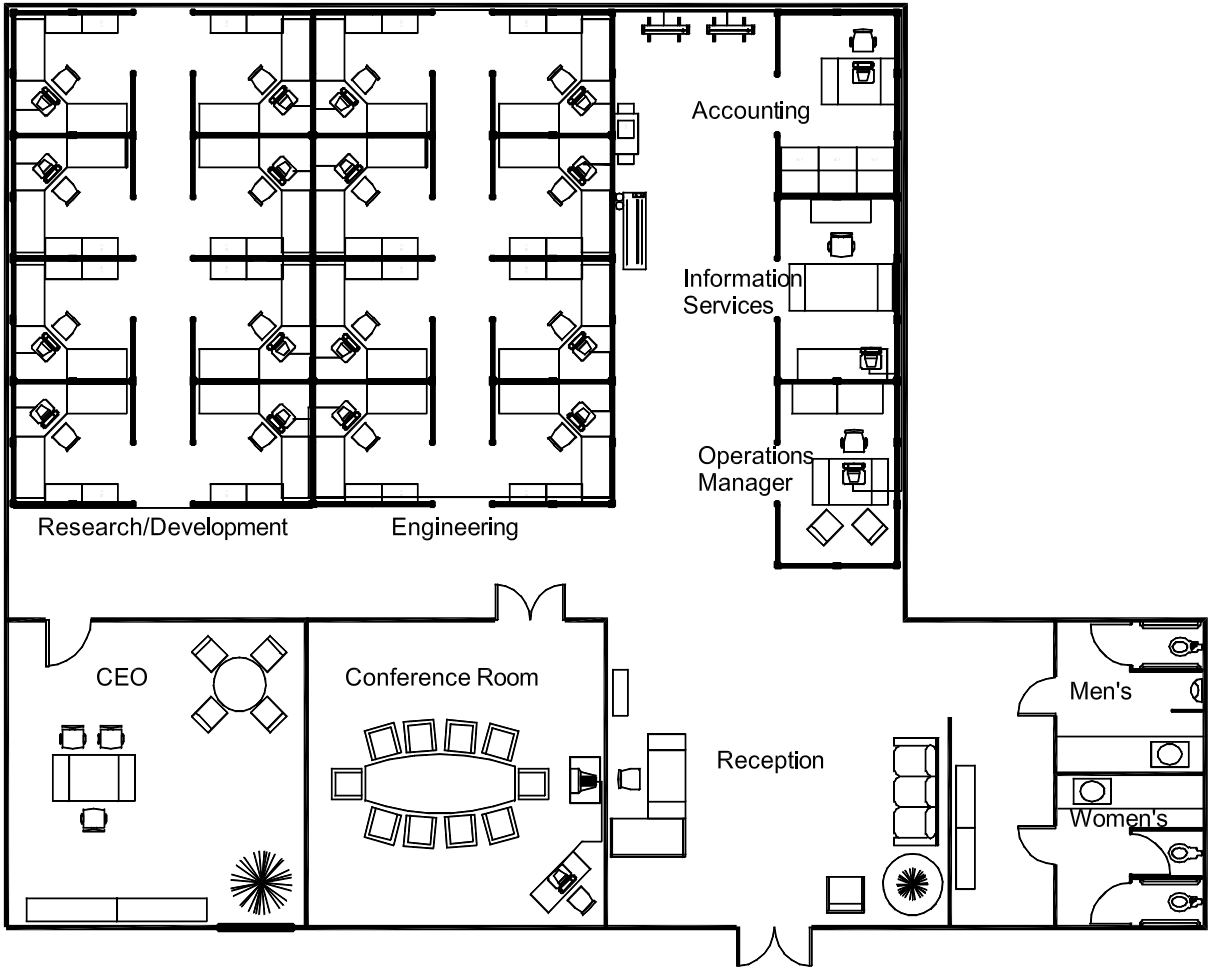
Jig – created using the Mechanical Part wizard



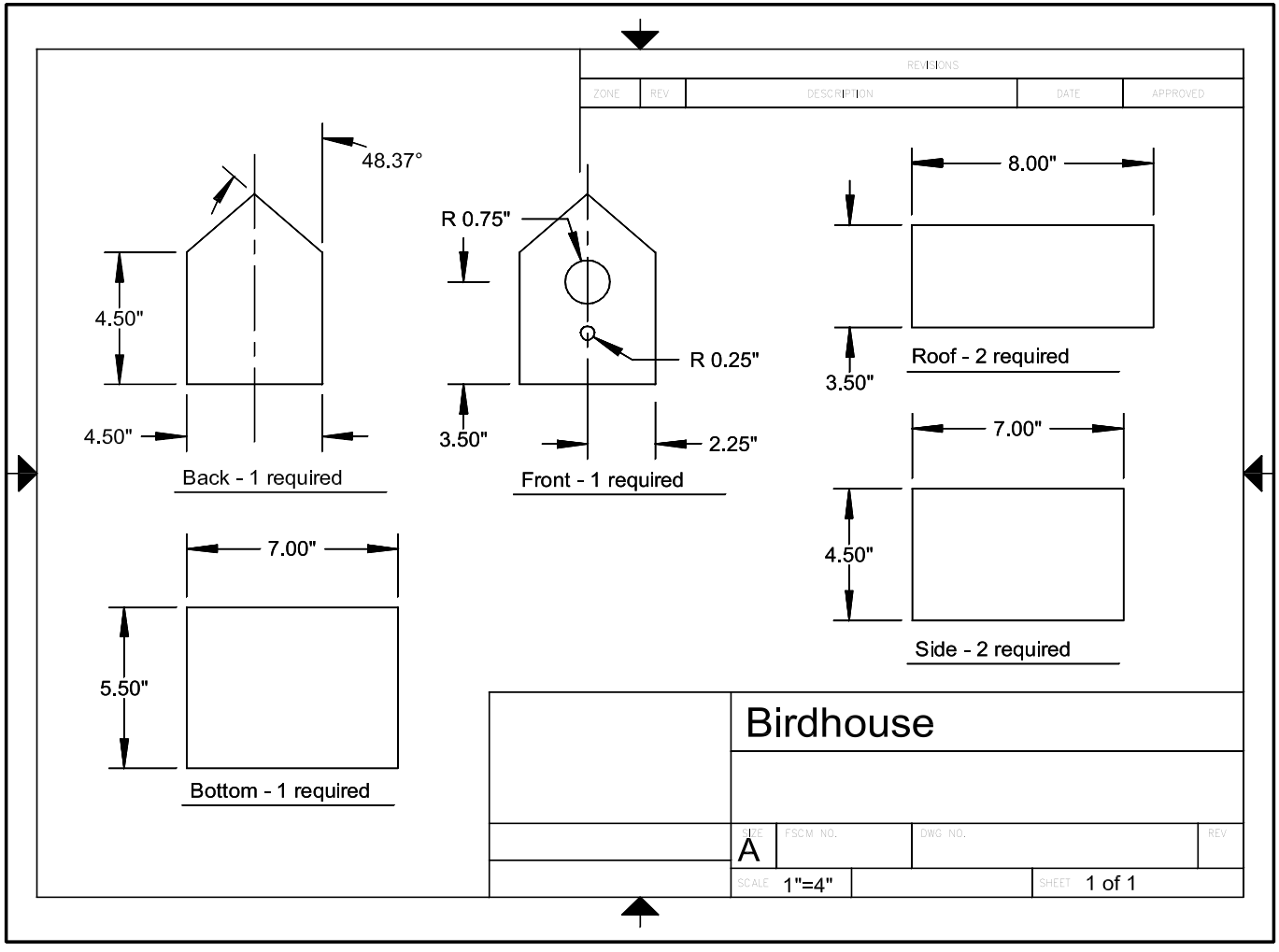
Test Site – created using the Diagram wizard



Tuner Diagram – created using the Diagram wizard



Utopia – Created using the Building wizard



Birdhouse – created using the Workbench wizard

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